2

WRDC-TR-89-2127

TURBINE ENGINE COMPONENT ANALYSIS: CANTILEVERED COMPOSITE FLAT PLATE ANALYSIS

W. R. Braisted
R. A. Brockman
J. W. Fielman

Structures Group, Aerospace Mechanics Division
University of Dayton Research Institute
Dayton, Ohio 45469

November 1989

Interim Report for Period November 1988 to March 1989

Approved for public release; distribution is unlimited.

AERO PROPULSION & POWER DIRECTORATE
WRIGHT LABORATORY
AIR FORCE SYSTEMS COMMAND
WRIGHT-PATTERSON AIR FORCE BASE, OHIO 45433-6563

NOTICE

WHEN GOVERNMENT DRAWINGS, SPECIFICATIONS, OR OTHER DATA ARE USED FOR ANY PURPOSE OTHER THAN IN CONNECTION WITH A DEFINITELY GOVERNMENT-RELATED PROCUREMENT, THE UNITED STATES GOVERNMENT INCURS NO RESPONSIBILITY OR ANY OBLIGATION WHATSOEVER. THE FACT THAT THE GOVERNMENT MAY HAVE FORMULATED OR IN ANY WAY SUPPLIED THE SAID DRAWINGS, SPECIFICATIONS, OR OTHER DATA, IS NOT TO BE REGARDED BY IMPLICATION, OR OTHERWISE IN ANY MANNER CONSTRUED, AS LICENSING THE HOLDER, OR ANY OTHER PERSON OR CORPORATION; OR AS CONVEYING ANY RIGHTS OR PERMISSION TO MANUFACTURE, USE OR SELL ANY PATENTED INVENTION THAT MAY IN ANY WAY BE RELATED THERETO.

THIS REPORT HAS BEEN REVIEWED BY THE OFFICE OF PUBLIC AFFAIRS (ASD/PA) AND IS RELEASABLE TO THE NATIONAL TECHNICAL INFORMATION SERVICE (NTIS). AT NTIS IT WILL BE AVAILABLE TO THE GENERAL PUBLIC INCLUDING FOREIGN NATIONS.

THIS TECHNICAL REPORT HAS BEEN REVIEWED AND IS APPROVED FOR PUBLICATION.

Daniel E Thomson

DANIEL E. THOMSON
Project Engineer
Components Branch
Turbine Engine Division

John A Vonada

JOHN A. VONADA, Lt Col, USAF
Chief
Components Branch
Turbine Engine Division

FOR THE COMMANDER

Richard J Hill

RICHARD J. HILL
Acting Deputy for Technology
Turbine Engine Division
Aero Propulsion & Power Directorate

IF YOUR ADDRESS HAS CHANGED, IF YOU WISH TO BE REMOVED FROM OUR MAILING LIST, OR IF THE ADDRESSEE IS NO LONGER EMPLOYED BY YOUR ORGANIZATION PLEASE NOTIFY WL/POTC, WRIGHT-PATTERSON AFB, OH 45433-6563 TO HELP MAINTAIN A CURRENT MAILING LIST.

COPIES OF THIS REPORT SHOULD NOT BE RETURNED UNLESS RETURN IS REQUIRED BY SECURITY CONSIDERATIONS, CONTRACTUAL OBLIGATIONS, OR NOTICE ON A SPECIFIC DOCUMENT.

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
89 NOV

3. REPORT TYPE AND DATES COVERED
Interim Nov 88 to Mar 89

4. TITLE AND SUBTITLE
Turbine Engine Component Analysis
Cantilivered Composite Flat Plate Analysis

5. FUNDING NUMBERS
F33615-87-C-2770

6. AUTHOR(S)
WR Braisted
RA Brockman
JW Fielman

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
University of Dayton Research Institute
300 College Park
Dayton, OH 45469-0001

8. PERFORMING ORGANIZATION REPORT NUMBER
UDR-TR-89-42

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Wright Research and Development Center
Aero Propulsion and Power Laboratory
Wright-Patterson AFB, OH 45433-6563

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
WRDC-TR-89-2127

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for Public Release; distribution is unlimited.

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The subject effort, "Turbine Engine Component Analysis", has as its objective the development and demonstration of in-house capabilities for computerized structural analyses of turbine engine components. Phase II, Structural Analysis, consists of a series of eight Tasks. This report describes work performed under Task 1, "Cantilevered Composite Flat Plate Analysis." The selected structure, a cantilevered composite flat plate subjected to a pressure load, is ideally suited for meeting the Task primary objective which is to demonstrate the plate and shell analysis capabilities of three computer codes used for engine blade-type structure. A second objective is to ensure that the various analysis programs, file transfer procedures, and data manipulation codes are fully operational and compatible. Twelve plate configurations are analyzed using three different finite element analysis codes: ADINA, COSMIC NASTRAN, and MAGNA. Results are compared with theoretical values to verify the finite element codes and the modeling assumptions.

14. SUBJECT TERMS
Computer graphics, structural analysis, finite elements.

15. NUMBER OF PAGES
64

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UNLIMITED

FOREWORD

The work described herein was performed between November 1988 and March 1989 at the University of Dayton Research Institute (UDRI), Dayton, Ohio. This task, "Cantilevered Composite Flat Plate Analysis", is a portion of the program conducted under U. S. Air Force Contract F33615-87-C-2770 for the Air Force Wright Aeronautical Laboratories Aero Propulsion and Power Laboratory (AFWAL/POT), Wright-Patterson Air Force Base, Ohio.

Technical direction and support for this effort were provided by Messrs. Daniel Thomson and Ted Fecke of AFWAL/POTC. The work was conducted within the Structures Group (Blaine S. West, Group Leader) of the Aerospace Mechanics Division (Dale H. Whitford, Project Supervisor). The UDRI Principal Investigator was Mr. John W. Fielman.

The authors also wish to acknowledge the efforts of Messrs. Thomas W. Held of UDRI and Les Whitford of the Air Force Aeronautical Systems Division, who provided technical and computer support for this task.

TABLE OF CONTENTS

SECTION 1

INTRODUCTION

The subject effort, "Turbine Engine Component Analysis", has as its objective the development and demonstration of in-house capabilities for computerized structural analyses of turbine engine components. Phase II, Structural Analysis, consists of a series of eight Tasks. The individual Tasks progressively become more complex and technically challenging. The structures to be analyzed for future Tasks will be selected to represent actual engine components. These analyses will be directed to produce results that are supportive of current Air Force engine design/development efforts.

This report describes the work performed under Task 1, "Cantilevered Composite Flat Plate Analysis." The selected structure, a cantilevered composite flat plate subjected to a pressure load, is ideally suited for meeting the Task primary objective which is to demonstrate the plate and shell analysis capabilities of the three computer codes used for engine blade-type structure. A second objective is to ensure that the various analysis programs, file transfer procedures, and data manipulation codes are fully operational and compatible. Twelve plate configurations are analyzed using three different finite element analysis codes: ADINA, COSMIC NASTRAN, and MAGNA. Results are compared with theoretical values to verify the finite element codes and the modeling assumptions.

SECTION 2

FINITE ELEMENT ANALYSIS CYCLE

As this is the first attempt to complete all the stages in a finite element analysis using the Air Force Aero Propulsion Laboratory facility, this document describes the specific aspects of model generation, job submission, data transfer, and results evaluation in great detail. The report will serve as an introductory user's guide for finite element analyses initiated and completed on the POTCM2 system.* Figure 1 depicts the fundamental steps of the finite element analysis cycle. The various computer systems and data communication pathways necessary to complete the finite element analysis are indicated in Figure 2. The accounts which follow describe each stage in the analysis cycle.

On POTCM2, geometry and finite element data are generated in PATRAN which is a widely used commercial finite element mesh generation and post-processing software package. The PATRAN model is stored in a PATRAN neutral file which is a formatted (ASCII) file written in a precise, well-documented format. The neutral file is PATRAN's communication link to other computer programs and computer systems.

The PATRAN pre-analysis translators convert the neutral file data into finite element input data for particular finite element analysis codes. The convention used to name the translators indicates both the direction of the translation and the computer codes involved. For example, PATADI converts PATRAN neutral file data into an ADINA input data deck. The post-analysis translator ADIPAT converts ADINA results into PATRAN results files.

For some analysis programs, an additional file is created during the pre-translation which establishes the correspondence between the PATRAN model and the finite element model. This file, referred to as the "saveme" file, will be required in post-processing when results are converted back into

---

* POTCM2 is the designation for the MicroVAX workstation installed at the Aero Propulsion Laboratory.

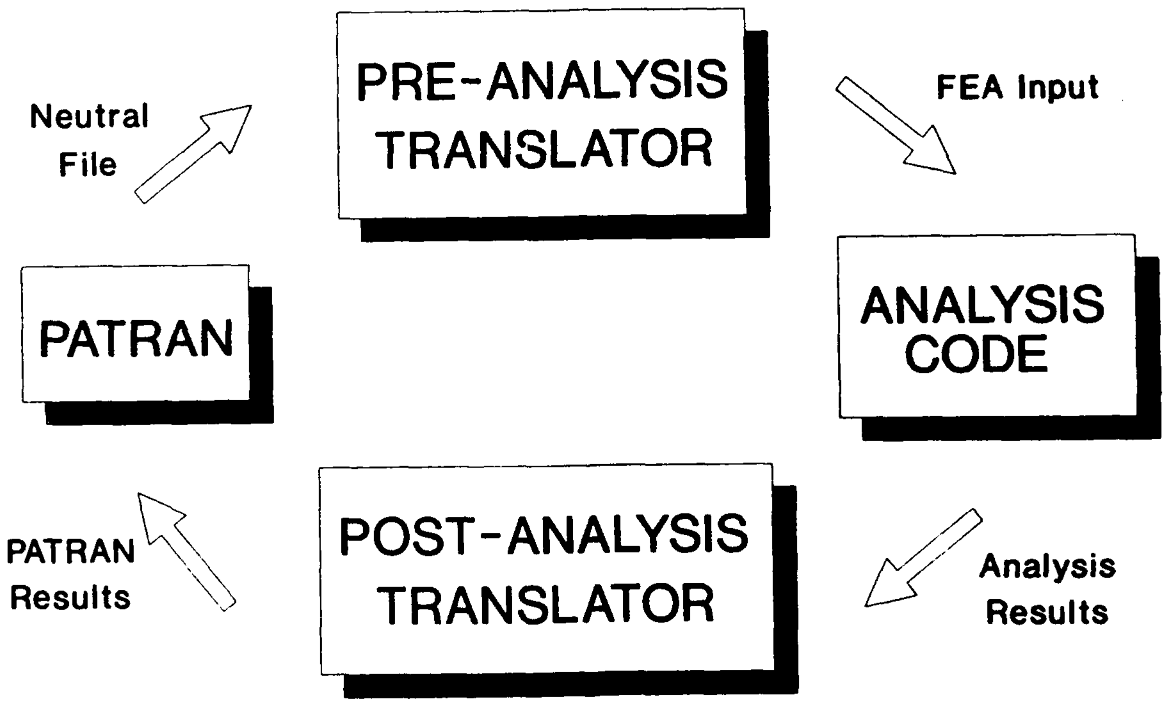


Figure 1. General Finite Element Analysis Cycle

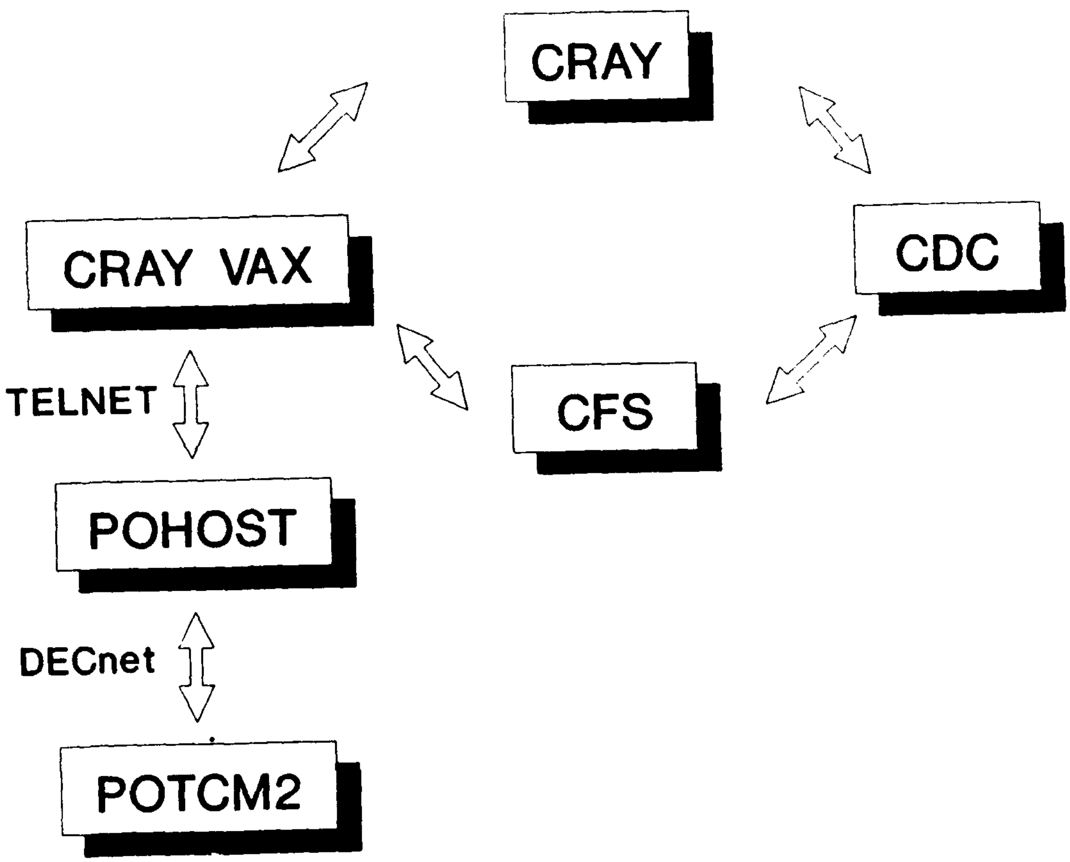


Figure 2. Computer Network Overview

PATRAN compatible formats. Several of the more widely used pre-analysis translators can be executed during a PATRAN session on the color graphics workstation (CGWS) or interactively at the VAX/VMS Operating System prompt ("$").

Analyses are to be performed on the CRAY to take advantage of the high speed and large memory capabilities of that machine. Unfortunately, no direct connection exists between the CRAY and POTCM2. Data files are transferred to the CRAY VAX, a front-end machine for the CRAY, using a combination of DECnet and TELNET commands. The communication link consists of DECnet protocols from POTCM2 to POHOST (ULTRIX system) and TELNET directives on the ARPANET system from POHOST to the CRAY VAX. Appendix A describes the DECnet/TELNET file transfer procedure.

Once the finite element input data is stored on the CRAY VAX, a batch job is submitted to the CRAY to perform the analysis. Section 5 describes the commands necessary to perform an analysis and to retrieve the resulting data. Due to the limited disk space on the CRAY VAX, results files from the analyses on the CRAY are written to the Central File System (CFS) by way of the CDC Cyber. Appendix B describes the procedures which are necessary to insure that CFS files are accessible on both the CDC and CRAY VAX.

Results stored on the CFS can be returned to POTCM2 in two ways. If the files are small enough to reside on the CRAY VAX, then they can be stored on disk on the CRAY VAX and retrieved on POTCM2 using the DECnet/TELNET commands in Appendix A. For larger analyses the CFS files must be read from the CDC and written to magnetic tape. This limitation was anticipated and is one of the major reasons why a tape drive was purchased for POTCM2. The tape is written with fixed length, blocked records such that the data can be read on POTCM2. Appendix C describes the recommended procedure to transfer data between the CDC and POTCM2 using magnetic tapes.

Post-analysis translation is performed to convert finite element output results data files into PATRAN results files. In most cases, the translation is executed on POTMC2 after the results are transferred from the CRAY. For COSMIC NASTRAN, PATRAN results files are created on the CRAY and then transferred to POTCM2.

Finally, results can be processed and displayed graphically in PATRAN on the CGWS. Several graphical methods are available to display results including deformed geometry plots and stress/strain contour plots. Hardcopy images can be produced on the thermal wax Tektronix 4693D Color Image Printer or on the LN03 laser printer (see Appendix D).

SECTION 3

PROBLEM DEFINITION

The structure under consideration is a cantilevered flat plate subjected to a uniform pressure load as indicated in Figure 3. The plate consists of 0.005 inch thick Ti_3Al/SCS-6 laminate whose material properties are presented in Figure 4. The silicone carbide fibers in each ply are unidirectional and the plies are stacked such that all the fibers align with the Y (longitudinal) axis of the model. A static pressure of 1.0 psi is applied to the entire surface. Table 1 shows the 12 geometry configurations evaluated for Task 1. Analyses of all twelve cases were performed using three finite element codes: ADINA[1], COSMIC NASTRAN[2], and MAGNA[3].

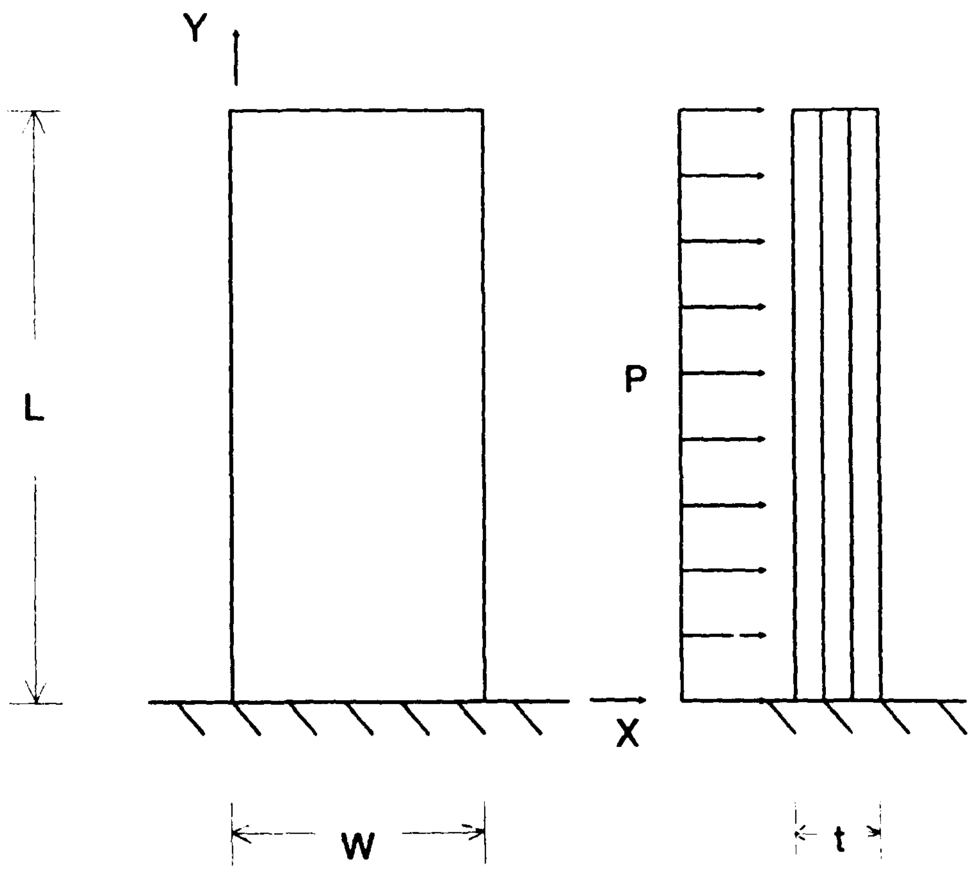
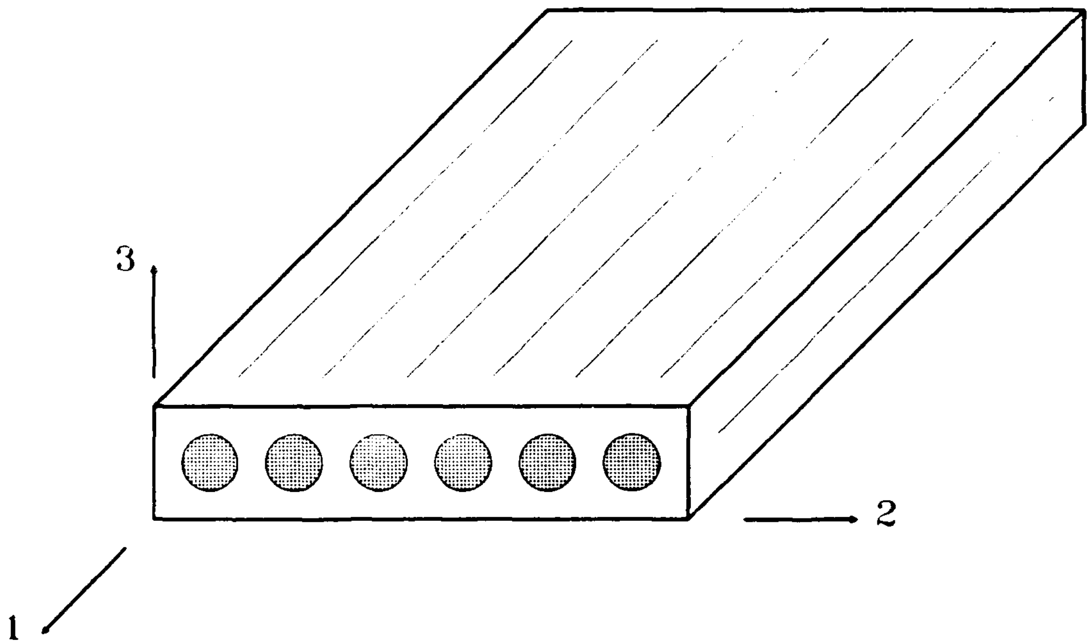


Figure 3. Task 1 Cantilevered Plate Geometry



ρ = 0.000381 lb_f-s^2/in^4		
E_1 = 28 Mpsi	E_2 = 14 Mpsi	E_3 = 14 Mpsi
G_{12} = 5.3 Mpsi	G_{13} = 5.3 Mpsi	G_{23} = 5.3 Mpsi
ν_{12} = 0.27	ν_{13} = 0.27	ν_{23} = 0.35
α_1 = 3.78 x $10^{-6}/^oF$	α_2 = 6.98 x $10^{-6}/^oF$	α_3 = 6.98 x $10^{-6}/^oF$

Figure 4. Ti_3Al/SCS-6 Orthotropic Material Properties

Table 1

Task 1 Cantilevered Plate Configurations

Case Number	Number of Plies	Length (in)	Width (in)
1	3	2.	1.
2	5	2.	1.
3	7	2.	1.
4	3	2.	2.
5	5	2.	2.
6	7	2.	2.
7	3	4.	2.
8	5	4.	2.
9	7	4.	2.
10	3	4.	4.
11	5	4.	4.
12	7	4.	4.

SECTION 4

MODEL GENERATION

On POTCM2, PATRAN[4] is being used to automate the mesh generation and data reduction processes. Models constructed in PATRAN are transferred to other analysis programs with a neutral file which records the model geometry and properties in a precise ASCII format. A pre-analysis translator converts neutral file information into input data for a particular finite element code. A post-analysis translator converts the finite element results back into PATRAN compatible formats.

Model generation in PATRAN is divided into two phases. Phase 1 directives define the geometry using grid points, lines (1-D), patches (2-D), and hyperpatches (3-D). Phase 2 commands construct finite element data (nodes, elements, loads, boundary conditions, etc.) from the Phase 1 geometry. Most Phase 2 directives end with the characters "FEG" which indicate Finite Element Generation (e.g. "CFEG"- connectivity finite element generation). For more detail regarding any PATRAN commands or capabilities see Reference 4.

The twelve configurations analyzed here are ideally suited for parametric modeling. Since PATRAN accepts geometric data either from the keyboard or from a data file, a common file can be formed for all the configurations. This file is then modified slightly to account for the particular parameter values of each case.

The sections which follow describe the particular features of modeling the Task 1 geometry for analyses with ADINA, COSMIC NASTRAN, and MAGNA.

4.1 ADINA MODELING

The PATRAN directives required to generate the Case 1 ADINA finite element model are presented below. A single patch is constructed from two lines [7] and paved with finite element nodes [8]. From the 121 nodes defined during paving, 100 four node quadrilateral elements are defined [9]. The correspondence between the PATRAN and ADINA element types is established in PATADI, the pre-analysis translator for ADINA. The PATADI documentation[5] describes the ADINA data types (elements, loads, materials, etc.) supported by the translator and the corresponding PATRAN directives required during model creation.

```
GR,1,,0/0/0                    [1]
GR,2,,1/0/0                    [2]
GR,3,,1/2/0                    [3]
GR,4,,0/2/0                    [4]
LI,1,2G,,1,2                   [5]
LI,2,2G,,3,4                   [6]
PA,1,2L,,1,2                   [7]
GF,P1,,11/11                   [8]
CF,P1,QUAD/4/5,,1              [9]
PF,P1,QUAD/4/5,,1/0.015,1     [10]
DF,P1,PRES,0/0/-1,1           [11]
DF,P1,DISP,6(0),999,ED4       [12]
```

The element type selected for this problem was the PATRAN QUAD/4/5 element which translates into the ADINA shell element (Type 7) with thickness correction. PATADI automatically generates midsurface normal vectors for each node referenced by a shell element. Using thickness correction, the element thickness will be oriented along the midsurface direction. If no thickness correction is desired, the QUAD/4/6 elements should be used.

Even though PATRAN can generate generic element types (QUAD, TRI, etc.), it is usually necessary to select the analysis program and the element type prior to generating elements in PATRAN. The element subtype specified in PATRAN determines the specific element type used in the analysis program.

Thicknesses for all of the QUAD/4/5 elements in the patch are defined as 0.015 inches [10], which is the total thickness of the three ply laminate for Case 1. The uniform pressure load of 1.0 psi is applied to all the elements of patch 1 normal to the surface (-Z direction) of the model [11]. Positive pressures in ADINA input data are defined in the inward normal direction of the element face.

Boundary conditions at the fixed end of the cantilevered plate are specified using the DFEG directive for prescribed displacements [12].

Unfortunately, zero prescribed displacements are not always converted to boundary conditions by PATADI. Only zero prescribed displacements in set 999 translate into boundary conditions. The difference between zero prescribed displacements and boundary conditions is unimportant to the results of the problem, but does affect the time and cost of the solution. Boundary conditions eliminate certain degrees of freedom from the system of equilibrium equations. Prescribed displacements create forces which influence the response of the unconstrained degrees of freedom. Matrix Equation 1 defines the equilibrium relations with prescribed displacements.

$$\begin{bmatrix} \mathbf{K}_{ff} & \mathbf{K}_{fp} \\ \mathbf{K}_{pf} & \mathbf{K}_{pp} \end{bmatrix} \begin{bmatrix} \mathbf{X}_{f} \\ \mathbf{X}_{p} \end{bmatrix} = \begin{bmatrix} \mathbf{F}_{f} \\ \mathbf{F}_{p} \end{bmatrix} \tag{1}$$

where:

f = free degrees of freedom

p = prescribed degrees of freedom

The solution to this system of equations requires two numerical operations as defined in Equations 2 and 3.

$$\mathbf{X}_{f} = \mathbf{K}_{ff}^{-1} \left[\mathbf{F}_{f} - \mathbf{K}_{fp} \mathbf{X}_{p} \right] \tag{2}$$

$$\mathbf{F}_{p} = \mathbf{K}_{pf} \mathbf{X}_{f} + \mathbf{K}_{pp} \mathbf{X}_{p} \tag{3}$$

In the first operation, the unknown displacements are determined. The product ($\mathbf{K}_{fp}$ $\mathbf{X}_{p}$) indicates the force at the unconstrained degrees of freedom required to produce the prescribed displacements. Even though the term is trivial when zero prescribed displacements are applied, all the calculations still are performed by ADINA. The second stage calculates the forces required to produce the prescribed displacements.

PATADI converts the PATRAN neutral file information into an ADINA input data file. Execution of PATADI can occur as a spawned process during a PATRAN session or externally in the VMS shell by typing:

$ RUN SYS1:[PATRAN.INTERFACE.PATADI]PATADI

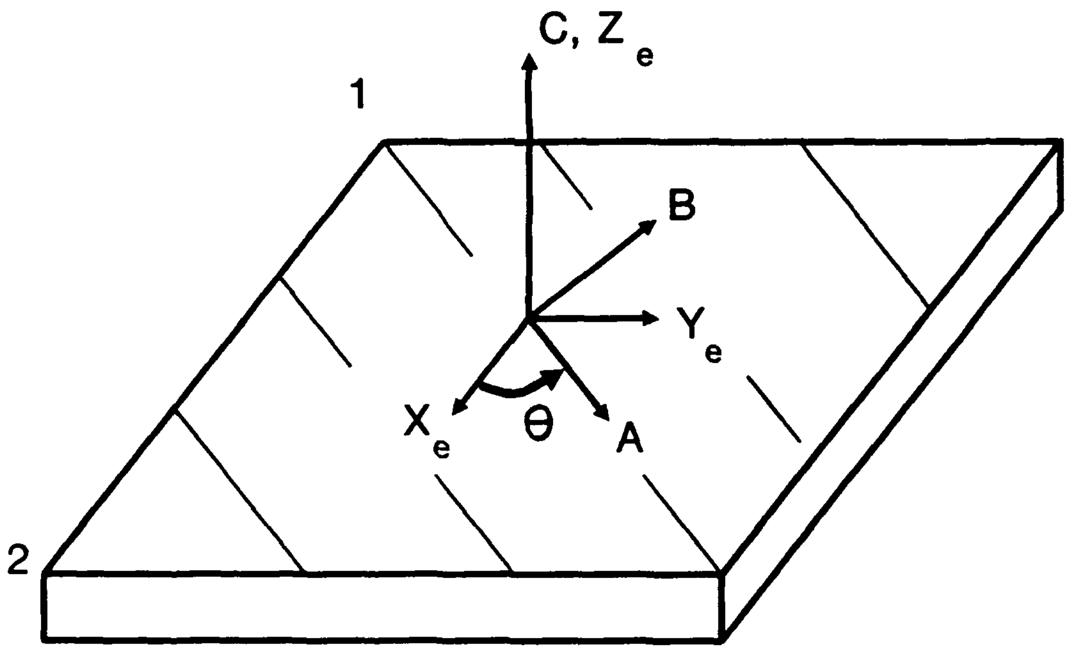
Specifying the formatted results option ensures that the ADINA analysis results can be read by the post-analysis translator ADIPAT.

No material property information is defined in the PATRAN commands because PATADI only supports linear elastic, isotropic materials for shell elements. After PATADI executes, the orthotropic material properties of the laminate must be entered into the data manually. This consists of changing the material model number on the element control card and specifying the constitutive properties. The orthotropic stress-strain relation for the ADINA shell element is:

$$\begin{bmatrix} \sigma_{aa} \\ \sigma_{bb} \\ \tau_{ab} \\ \tau_{ac} \\ \tau_{bc} \end{bmatrix} = \begin{bmatrix} Q_{aa} & Q_{ab} & 0 & 0 & 0 \\ Q_{ab} & Q_{bb} & 0 & 0 & 0 \\ 0 & 0 & G_{ab} & 0 & 0 \\ 0 & 0 & 0 & G_t & 0 \\ 0 & 0 & 0 & 0 & G_t \end{bmatrix} \begin{bmatrix} \epsilon_{aa} \\ \epsilon_{bb} \\ \gamma_{ab} \\ \gamma_{ac} \\ \gamma_{bc} \end{bmatrix} \qquad (4)$$

in which (a,b) are the principal material axes parallel to the shell surface, and c is the mid-surface normal direction. Note that the meaning of the material directions (a,b,c) is tied to the element connectivity data (Figure 5). The material orientation angle θ defines the fiber direction with respect to the 1-2 element edge. Therefore changing the order of the connectivity also may require a change in the material orientation angle. PATRAN generates the connectivity automatically, so the material definition should always be checked to insure the material model is aligned properly.

In general, it is necessary to determine the orientation of the connectivity generated by PATRAN in order to verify that the orthotropic properties are prescribed properly. The ordering of the connectivity usually defines the element local axes, which in turn are used to specify material fiber orientations. The conventions for local axis and material axis definitions differ between programs, and often between element types in a single program.



X_e, Y_e, Z_e = element coordinate system
(X_e parallel to the line segment from node 1 to node 2)
A, B, C = axes of material orthotropy

θ = material orientation angle

$$\begin{bmatrix} A \\ B \\ C \end{bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta & 0 \\ -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} X_e \\ Y_e \\ Z_e \end{bmatrix}$$

Figure 5. Orthotropic Material Axis Definition

The independent mechanical properties are the extensional moduli E_a, E_b, the major Poisson's ratio ν_{ab}, the inplane shear modulus G_{ab}, and the transverse shear modulus G_t. The components of the constitutive relation in the material coordinate system are calculated directly from the orthotropic material properties using[6]:

$$Q_{aa} = E_a / (1 - \nu_{ab}\nu_{ba}) \tag{5}$$

$$Q_{ab} = \nu_{ab}E_b / (1 - \nu_{ab}\nu_{ba}) \tag{6}$$

$$Q_{bb} = E_b / (1 - \nu_{ab}\nu_{ba}) \tag{7}$$

where:

$$\nu_{ba} = E_b \, \nu_{ab} / E_a \tag{8}$$

$$| Q_{ab} | < (Q_{aa} Q_{bb})^{1/2} \tag{9}$$

It should be noted the convention used above to define the minor Poisson's ratio ν_{ba} is the standard.[11] However, the ADINA documentation uses the opposite definition. Thus, when ADINA requires ν_{ab}, the correct input is ν_{ba} using the normal definition. Care must be used when specifying or collecting material properties to follow the convention used. Inappropriate designation of the Poisson's ratios can result in a constitutive relation which is not positive definite and thereby numerically ill-conditioned.

4.2 COSMIC NASTRAN MODELING

PATRAN directives for the COSMIC NASTRAN Case 1 model are presented below. Due to the similarity of the PATRAN commands for the ADINA and COSMIC NASTRAN models, only the differences are discussed.

```
GR,1,,0/0/0                        [1]
GR,2,,1/0/0                        [2]
GR,3,,1/2/0                        [3]
GR,4,,0/2/0                        [4]
LI,1,2G,,1,2                       [5]
LI,2,2G,,3,4                       [6]
PA,1,2L,,1,2                       [7]
GF,P1,,11/11                       [8]
CF,P1,QUAD//,,1                    [9]
PF,P1,QUAD//,,1/0.015,1            [10]
DF,P1,PRES,0/0/-1,1                [11]
DF,P1,DISP,6(0),2,ED4              [12]
```

The appropriate COSMIC NASTRAN element for the Task 1 analyses is the QUAD4 element. The QUAD4 element is preferred over the QUAD2 element for general use because of some numerical instabilities which can occur using the QUAD2 element. Additionally, offsets from the midplane can be specified for each of the four corner nodes using the QUAD4 element.

Unfortunately, the QUAD4 element was incorporated into COSMIC NASTRAN (1988 version) after the last versions of PATCOS and COSPAT (1984). Thus, models requiring the QUAD4 element cannot be created using PATRAN and PATCOS exclusively. The solution is to generate QUAD2 type information in PATRAN, which can be translated, and then manually edit the COSMIC NASTRAN input file to account for the QUAD4 data. The conversion to the QUAD4 element requires three basic modifications.

First, the CQUAD2 records defining the element connectivity must be replaced with the corresponding CQUAD4 cards. Second, the PQUAD2 card must be replaced. The PSHELL card is the corresponding card (no PQUAD4 card exists), but it must be used with caution. Independent material identification numbers can be specified for the membrane, bending, transverse shear, and coupling properties. Certain restrictions are indicated in the documentation regarding acceptable material types for transverse shear properties, but no error messages occur in the output if some of these restrictions are violated. Third, the material properties must be specified. The appropriate constitutive relation is a plane stress material with transverse shear stiffness (MAT8). The MAT8 card is available in the latest version of COSMIC NASTRAN, but is not supported by PATCOS. All MAT8 references must be added manually to the COSMIC NASTRAN input data deck. Appendices E, F, and G contain information regarding the format and the data requirements of the CQUAD4, PSHELL, and MAT8 cards for recent versions of COSMIC NASTRAN[8].

It is important to note that the orientation of the material properties is tied to the element connectivity data. Changing the order of the connectivity also may require a change in the material definition. PATRAN generates the connectivity automatically, so the material description should always be checked to insure proper alignment.

The bulk data generated by PATCOS represents the majority, but not all, of the COSMIC NASTRAN input data. PATCOS does not create any of the Executive and Case Control decks required by NASTRAN. Often a file with typical Executive and Case Control directives can be used as a template for future analyses. The Executive and Case Control cards for the first configuration of the Task 1 effort presented below are typical of many COSMIC NASTRAN applications. Presented below, the Executive and Case Control cards for the first Task 1 configuration are typical of many COSMIC NASTRAN applications.

```
ID COSMIC,QUAD4                                    [1]
APP DISP                                           [2]
SOL 1,0                                            [3]
TIME 5                                             [4]
DIAG 14                                            [5]
ALTER 143                                          [6]
OUTPUT2 OUGV1,OES1//C,N,-1/C,N,11/V,N,Z $          [7]
ENDALTER                                           [8]
CEND                                               [9]
TITLE        = TASK 1   CASE 1                    [10]
SUBTITLE     = COSMIC NASTRAN                     [11]
DISPLACEMENT = ALL                                [12]
ECHO         = ALL                                [13]
ELFORCE      = ALL                                [14]
ELSTRESS     = ALL                                [15]
LOAD         = 1                                  [16]
OLOAD        = ALL                                [17]
SPC          = 2                                  [18]
SPCFORCE     = ALL                                [19]
```

Loads [16] and boundary conditions [18] in sets 1 and 2 respectively, must correspond to the set numbers used in PATRAN during model creation. If the identifications do not match, then the desired loads or boundary conditions will not be applied during the analysis execution. Pressure loads created in PATRAN (DFEG,,PRES) are converted in PATCOS into PLOAD2 cards. These loads can be applied to QUAD4 elements without difficulty. Also, the problems associated with zero prescribed displacements that occur in PATADI are not troublesome in PATCOS.

COSPAT requires COSMIC NASTRAN results from the binary OUTPUT2 file. The OUTPUT2 file is not generally created during COSMIC NASTRAN execution, so special DMAP (Direct Matrix Abstraction Program)[2] commands are included in the Executive Control deck [6-8]. These DMAP commands modify the predefined order in which the DMAP modules are invoked. The ALTER command indicates the location in the rigid format DMAP sequence after which the supplementary commands are added. This location depends upon the analysis type (NASTRAN rigid format number). The example shown above is correct for linear static analysis (SOL 1). Diagnostic 14 [5] is initiated to produce a complete listing of the DMAP commands invoked at execution. This is useful to verify that the DMAP ALTER was executed at the appropriate phase in the solution.

COSPAT documentation[7] indicates that von Mises stresses will be written to the PATRAN element results file if Diagnostic 33 is initiated in the Executive Control deck. DIAG 33 is an MacNeal-Schwendler Corporation (MSC) NASTRAN command to record effective stresses in the output data. No such command exists for COSMIC NASTRAN. If von Mises stresses are desired from a COSMIC NASTRAN analysis, then the program VMCOSMIC should be executed on POTCM2 with the original PATRAN element results file. To run VMCOSMIC type:

$ RUN USERA:[TASK21]VMCOSMIC

VMCOSMIC will calculate von Mises stresses and store them in column 32 of the PATRAN element results file. Effective stresses are calculated for PATRAN element types TRI, QUAD, TETRA, WEDGE, and HEX. All other element types will be assigned a zero value.

The OUTPUT2 file from a CRAY NASTRAN run is a CRAY binary file, and must be translated into PATRAN-readable form on the CRAY. Unfortunately, no CRAY version of COSPAT is available from PDA Engineering, the authors of PATRAN. A CDC version of COSPAT was obtained and converted to operate on the CRAY. During the port to the CRAY, provisions for translating QUAD4 element results were added to COSPAT. COSMIC NASTRAN execution on the CRAY is followed immediately by COSPAT execution. The PATRAN results files produced in COSPAT must be formatted (/ASCII:ON option)[7] if the files are to be returned to the CGWS for post-processing.

It should be noted that the conversion of COSPAT to the CRAY X-MP is not yet complete. Additional work is required to support other analysis options (such as normal modes), additional finite element types, and to handle multiple solution sets automatically. Appendix H summarizes the modifications that have been completed to date.

4.3 MAGNA MODELING

PATRAN directives for the MAGNA Case 1 model are presented below. The ADINA and COSMIC NASTRAN models are similar due to the fact that both use two-dimensional elements for the model. The appropriate MAGNA element type for Task 1 is the layered shell element (Type 11) shown in Figure 6. The layered shell element is a sixteen-node solid element consisting of multiple plies of different materials at unique orientations. The fully three-dimensional nature of the MAGNA layered shell element requires a more complex model than the ADINA and COSMIC NASTRAN counterparts.

```
GR,1,,0/0/0                                        [1]
GR,2,,1/0/0                                        [2]
GR,3,,1/2/0                                        [3]
GR,4,,0/2/0                                        [4]
LI,1,2G,,1,2                                       [5]
LI,2,2G,,3,4                                       [6]
PA,1,2L,,1,2                                       [7]
PA,2,TR,0/0/0.015,1                                [8]
HP,1,2P,,1,2                                       [9]
GF,H1,,11/11/2                                    [10]
CF,H1,HEX/20/7,,1                                 [11]
DATA,1,(3, 1,1,0.005,0,
           1,1,0.005,0,
           0,1,1.0,0   )                          [12]
PF,H1,HEX/20/7,D1                                 [13]
DF,H1,PRES,0/0/-1,F6                              [14]
DF,H1,DISP,6(0),999,F3                            [15]
PMAT,1,ORT,( 14.E6, 28.E6, 14.E6,
             0.135, 0.,    0.,
             0.000381, 3(5.3E6),
             3.78E-6, 6.98E-6, 6.98E-6 )          [16]
```

As was the case for the ADINA and COSMIC NASTRAN models a patch is constructed [7]. For the MAGNA model, a second patch is generated by translation of the first patch through the plate thickness [8]. The volume between the patches is converted into a hyperpatch [9] and paved with nodes [10].

When PATMAG[9] the pre-analysis translator for MAGNA was written no sixteen-node elements were supported in PATRAN. The method to create the

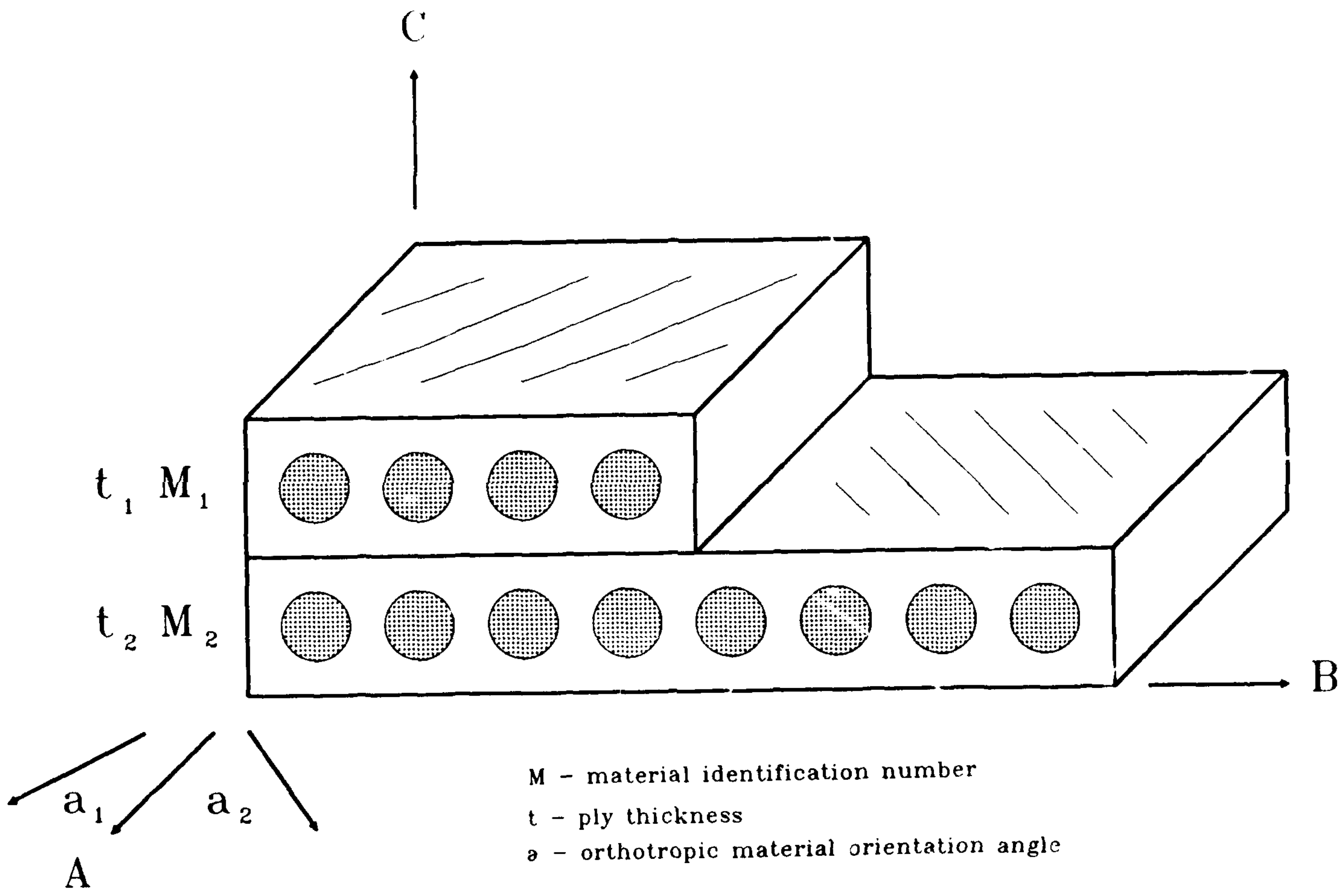


Figure 6. MAGNA Sixteen-node Layered Shell Element.

sixteen-node layered shell elements was to degenerate the twenty-node solid elements that PATRAN supported (HEX/20/7 element). Currently, PATRAN HEX/20/7 elements are still used even though sixteen-node solid elements are available in PATRAN. PATMAG and MAGPAT eliminate and restore the extraneous midside nodes. Results for the midside nodes are calculated by linear interpolation.

Using the HEX/20/7 elements, 803 nodes are generated defining the 100 elements of the model [11]. The 803 nodes is significantly larger than the 121 nodes in the ADINA and COSMIC NASTRAN meshes (note, however, that the number of degrees of freedom per node is three rather than six). In exchange for a larger and more detailed model (and higher computation costs), stress and strain information can be extracted for the upper and lower surfaces of each ply.

To describe the laminate composition, the element properties [13] reference a PATRAN DATA line [12]. The general DATA line format used to define layered shell elements is indicated below.

```
DATA,ID,( N, L(1), M(1), T(1), a(1),
             L(2), M(2), T(2), a(2),
              "     "     "     "
             L(N), M(N), T(N), a(N)  )
```

where:

N - number of layers in the element (maximum of 11)

ID - data line identification number

L(i) - layer type of layer "i"

=0 variable thickness OR

=1 constant thickness

M(i) - material identification number of layer "i"

T(i) - thickness of layer "i" (L(i)=1) OR

thickness fraction of layer "i" (L(i)=0)

a(i) - orthotropic material axis orientation angle of layer "i"

Note: 1. at least one layer in each element must be variable thickness

2. thickness fractions are percentages of the remaining thickness

3. see Reference 3 for further details

The constitutive relation for the MAGNA layered shell element is fully three-dimensional. The transverse longitudinal stiffness through the plate thickness was not modeled in the ADINA and COSMIC NASTRAN relations. For thin plates the transverse longitudinal stiffness should be uncoupled from the remaining stiffness terms. If a fully orthotropic material is defined in PATRAN, PATMAG will not uncouple the stiffness. To insure that this does occur the Poisson's ratios ν_{12} and ν_{13} defined in PATRAN are set to zero. The following matrix expression describes the resulting constitutive relationship.

$$\begin{bmatrix} \sigma_{aa} \\ \sigma_{bb} \\ \sigma_{cc} \\ \tau_{ab} \\ \tau_{ac} \\ \tau_{bc} \end{bmatrix} = \begin{bmatrix} Q_{aa} & Q_{ab} & 0 & 0 & 0 & 0 \\ Q_{ab} & Q_{bb} & 0 & 0 & 0 & 0 \\ 0 & 0 & E_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & G_{ab} & 0 & 0 \\ 0 & 0 & 0 & 0 & G_t & 0 \\ 0 & 0 & 0 & 0 & 0 & G_t \end{bmatrix} \begin{bmatrix} \epsilon_{aa} \\ \epsilon_{bb} \\ \epsilon_{cc} \\ \gamma_{ab} \\ \gamma_{ac} \\ \gamma_{bc} \end{bmatrix} \qquad (10)$$

It is important to note that the meaning of the material directions (a,b,c) is tied to the element connectivity data (Figure 5). The material orientation angle θ defines the fiber direction with respect to the 1-2 element edge. Therefore changing the order of the connectivity also may require a change in the material orientation angle. PATRAN generates the connectivity automatically, so the material definition should always be checked to insure the material model is aligned properly.

Many of the most commonly used finite element codes have pre- and post-analysis translators which can be run during PATRAN execution. The MAGNA translators are not currently included in this list and hence, must be evoked externally. The command to execute PATMAG on the POTCM2 system is:

$ RUN SYS1:[PATRAN.INTERFACE.PATMAG]PATMAG

SECTION 5

ANALYSIS CODE EXECUTION

Finite element analyses are performed on the CRAY to take advantage of the high speed and large memory capabilities of the machine. This section describes procedures for executing the analysis programs (ADINA, COSMIC NASTRAN, and MAGNA) on the CRAY X-MP at Wright Patterson Air Force Base.

A procedure file, CRAY_SUBMIT, was written to generate the necessary CRAY job control language to perform a finite element analysis on the CRAY X-MP at Wright Patterson Air Force Base. The procedure contains all the job control language (JCL) necessary to submit and to retrieve data from static or eigenvalue analyses using ADINA, COSMIC NASTRAN, or MAGNA. Analyses requiring restart data to be written or retrieved will necessitate slight editing of the JCL.

Results are generally returned to the same front-end machine from which they are submitted to the CRAY; however, the limited amount of CRAY VAX disk space may be insufficient to store the analysis results. To alleviate the problem, results can be routed to the Central File System (CFS) through the CDC Cyber. The file size limitations are not of practical concern with this approach. Smaller results files can be read from the CFS onto the CRAY VAX directly and retrieved on POTCM2 using DECnet/TELNET commands (Appendix A); larger results files are written to magnetic tape for retrieval (Appendix C). The CRAY report file detailing the analysis time and cost is returned to the CRAY VAX.

ADINA and MAGNA analysis submittal procedures fetch the input data from the CRAY VAX on execution. COSMIC NASTRAN input data is appended to the submittal procedure. The reason for the different methodology is the manner in which COSMIC NASTRAN is accessed and executed on the CRAY. To generate the CRAY JCL necessary to perform a CRAY analysis, type:

$ @[P890028]CRAY_SUBMIT

The CRAY JCL will be written to a file having the same name as the input data but with the extension .JOB. The resulting CRAY job submittal procedure for an ADINA analysis is presented in Table 2. Further information regarding the CRAY operating system and the available commands is contained in Reference 10.

Analysis results from the ADINA (Porthole file) and MAGNA (MPOST file) analyses are formatted (ASCII) files and can be returned to POTCM2 for subsequent post-processing. COSMIC NASTRAN results (OUTPUT2 file) are binary, and therefore highly machine dependent. COSPAT must be executed on the CRAY immediately following COSMIC NASTRAN analysis; the resulting formatted PATRAN results files are returned to POTCM2 (see Section 4.2). Note that COSMIC NASTRAN job submittal procedures written by CRAY_SUBMIT automatically execute COSPAT after the finite element analysis.

Table 2.

Representative CRAY Job Submittal Procedure

```
JOB,JN=adina,T=200,US=P890028,MFL.
ACCOUNT,AC=P890028,APW=crayapw,UPW=crayupw.
*.
*. CRAY JOB SUBMISSION
*. ADINA FINITE ELEMENT ANALYSIS
*.
REWIND,DN=$OUT.
*.
*. RETRIEVE INPUT FILE FROM CRAY VAX
*. AND ASSIGN LOCAL DATASETS
*.
FETCH,DN=ADDAT,SDN=ADDAT,TEXT='adina.dat'.
ASSIGN,DN=ADDAT,A=FT05.
ASSIGN,DN=PORTF,A=FT61.
*.
*. ACCESS AND EXECUTE ADINA
*.
ACCESS,DN=ADINA,PDN=ADINA84,OWN=D870062.
ADINA.
*.
*. ROUTE STANDARD OUTPUT AND PORTHOLE RESULTS TO THE CFS
*.
DISPOSE,DN=$OUT,SDN=OUT,MF=CB,DF=CB,DC=ST,^
TEXT='USER,P890028,cdcbpw.CTASK.FILE,OUT,BT=C,RT=Z,FL=132.'^
'FILE,AOUT,BT=C,RT=Z,FL=132.FORM,INP=OUT,OUT=AOUT.'^
'SWRITE,AOUT,adinaot,RFMT=VLB,STYP=DIS,TTYP=ASC.'.
DISPOSE,DN=PORTF,SDN=PORTF,MF=CB,DF=CB,DC=ST,^
TEXT='USER,P890028,cdcbpw.CTASK.FILE,PORTF,BT=C,RT=Z,FL=132.'^
'FILE,APORT,BT=C,RT=Z,FL=132.FORM,INP=PORTF,OUT=APORT.'^
'SWRITE,APORT,adinaph,RFMT=VLB,STYP=DIS,TTYP=ASC.'.
```

where:

adina.dat = CRAY VAX input data file

adinaot = Standard output file on the CFS

adinaph = Porthole results file on the CFS

crayapw = CRAY account password

crayupw = CRAY user password

cdcbpw = CDC batch password

To submit the CRAY JCL file listed above, type:

$ CRAY SUBMIT adina.job

SECTION 6

BEAM THEORY SOLUTION

As this is the first attempt to use all of the various computers, file transfer procedures, and programs involved in the complete analysis cycle, a problem was selected whose behavior was well known. Using standard beam theory relations,[11] the bending response of the plate can be estimated. The beam theory solution assumes that any variations along the width of the plate are minor compared to those along the length. For the geometry and load conditions in this Task, the beam theory solution provides first order approximate results which will aid in the evaluation of the finite element results and the modeling assumptions.

Figure 7 depicts an edge view of the cantilevered plate with a uniform distributed load. Integration of the applied pressure load over the plate surface area results in the expression for the bending moment (M) defined in Equation 11.

$$M(y) = - q \, (L - y)^2 / 2 \tag{11}$$

where:

y = distance along the plate length

L = plate length

q = $P \, d$ = load per unit length along y

P = pressure load over entire plate surface

d = plate width

Equation 12 relates the bending moment in the plate to the second derivative of the plate deflection (w).

$$- E \, I \, (\partial^2 w / \partial y^2) = M(y) \tag{12}$$

where:

I = area moment of inertia = $d \, t^3 / 12$

E = modulus of elasticity along y

t = plate thickness

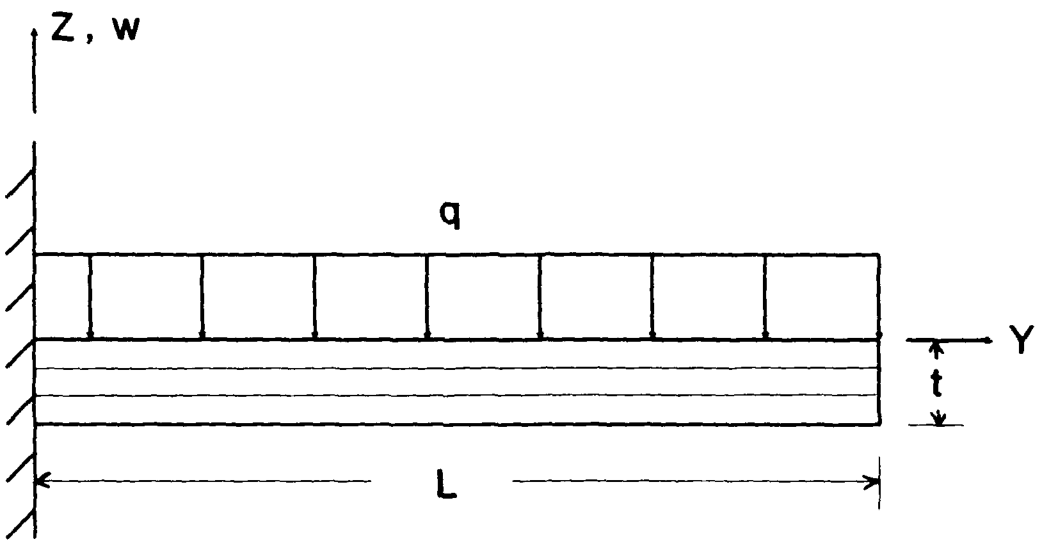


Figure 7. Edge View of the Task 1 Plate Geometry

Integration of the preceding equation with the appropriate boundary conditions results in Equation 13 for the deflection of the plate.

$$w(y) = -qy^2 \, (6L^2 - 4Ly + y^2) \, / \, 24EI$$

$$= -Py^2 \, (6L^2 - 4Ly + y^2) \, / \, 2Et^3 \qquad (13)$$

Note that the deflection is not a function of the plate width and varies with the fourth power of the length. Equation 14 defines the maximum displacement in the plate occurring along the free edge (y=L).

$$w_{max} = qL^4 \, / \, 8EI \; = \; 3PL^4 \, / \, 2Et^3 \qquad (14)$$

Assuming that the plate is subjected to pure bending, the tensile stress on the upper surface of the plate is related to the bending moment by Equation 15.

$$\sigma(y) = Mt \, / \, 2I \; = \; q \, (L-y)^2 \, t \, / \, 4I \; = \; 3 \, (L-y)^2 \, P \, / \, t^2 \qquad (15)$$

The maximum stress in the plate occurs along the support (y=0) and is defined in Equation 16.

$$\sigma_{max} = 3PL^2 \, / \, t^2 \qquad (16)$$

SECTION 7

RESULTS EVALUATION

The finite element analysis results must be translated into PATRAN results files. These results files contain information regarding the stress, strain, and displacement behavior at the nodes and elements of the model. No geometry information is included on the results files. A PATRAN neutral file is used to reconstruct the geometry upon which the finite element results are superimposed. PATRAN results files are of three types:

Element	- centroidal element stress and strain values
Nodal	- nodal stress and strain values
Displacement	- nodal displacements

PATRAN results files for COSMIC NASTRAN problems are generated by executing COSPAT on the CRAY immediately after the finite element analysis. ADINA and MAGNA finite element results are returned to POTCM2, and then converted to PATRAN compatible form.

After successful completion of a finite element analysis on the CRAY, the results required for post-processing are located on the Central File System (CFS). For COSMIC NASTRAN analyses, the files on the CFS are PATRAN results files because COSPAT is executed on the CRAY immediately after the COSMIC NASTRAN solution. For ADINA and MAGNA analyses, the finite element results files (ADINA Porthole and MAGNA MPOST) are written to the CFS. Data is returned to POTCM2 using either the DECnet/TELNET protocol (Appendix A) or magnetic tape (Appendix C). On POTCM2, Porthole and MPOST files must be converted to PATRAN results files using the respective post-analysis translators, ADIPAT and MAGPAT.

Post-processing in PATRAN consists of deformed geometry plots or equivalent stress and strain contour plots. Hardcopy can be generated on the Tektronix 4396D color printer or on the LN03 laser printer (see Appendix D). Figures 8 to 10 indicate the total displacement state in the plate for Case 1 ADINA, COSMIC NASTRAN, and MAGNA analyses, respectively. Similarly, von Mises stress contours for the Case 1 ADINA, COSMIC NASTRAN, and MAGNA

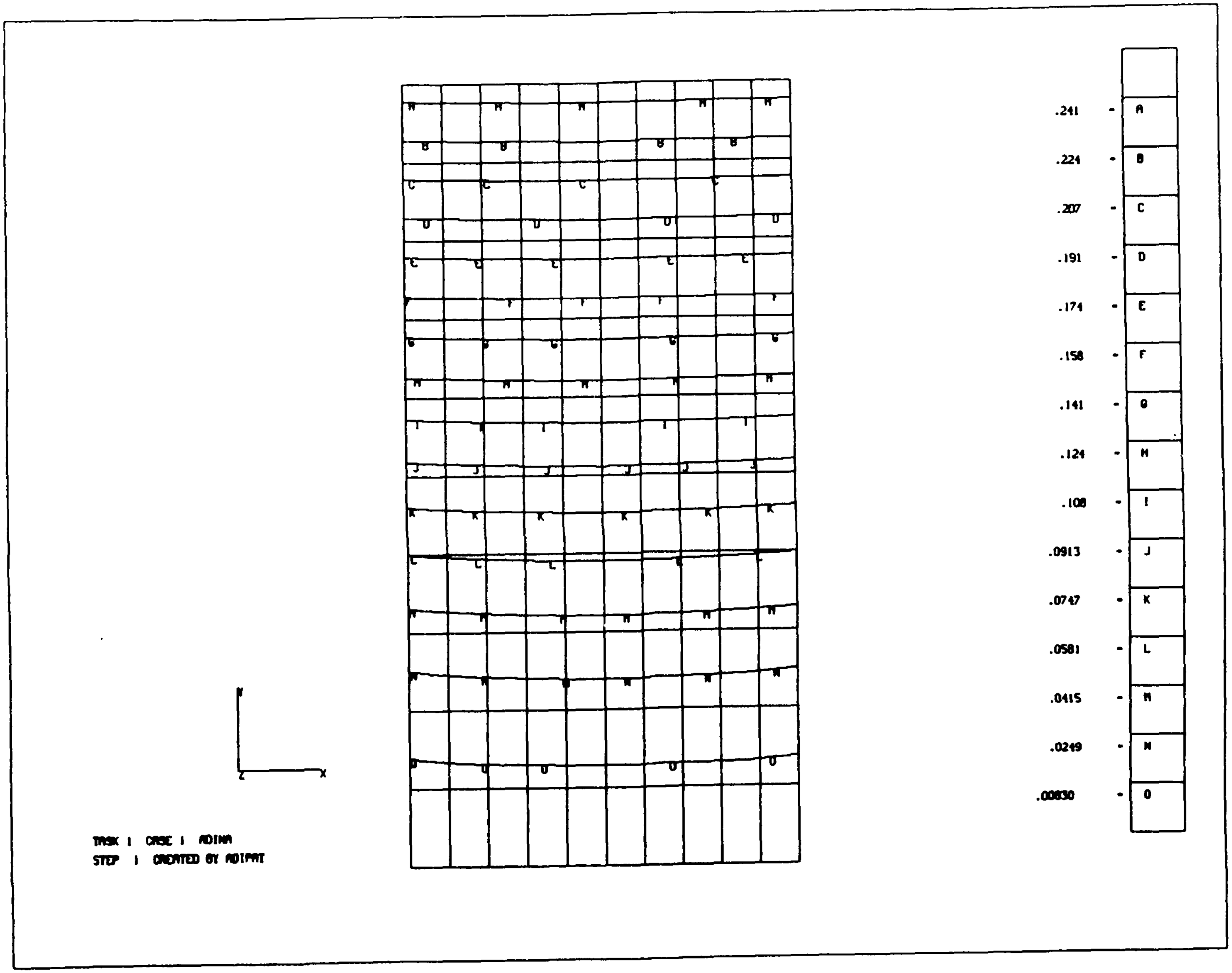


Figure 8. ADINA Equivalent Displacement Contours for Case 1.

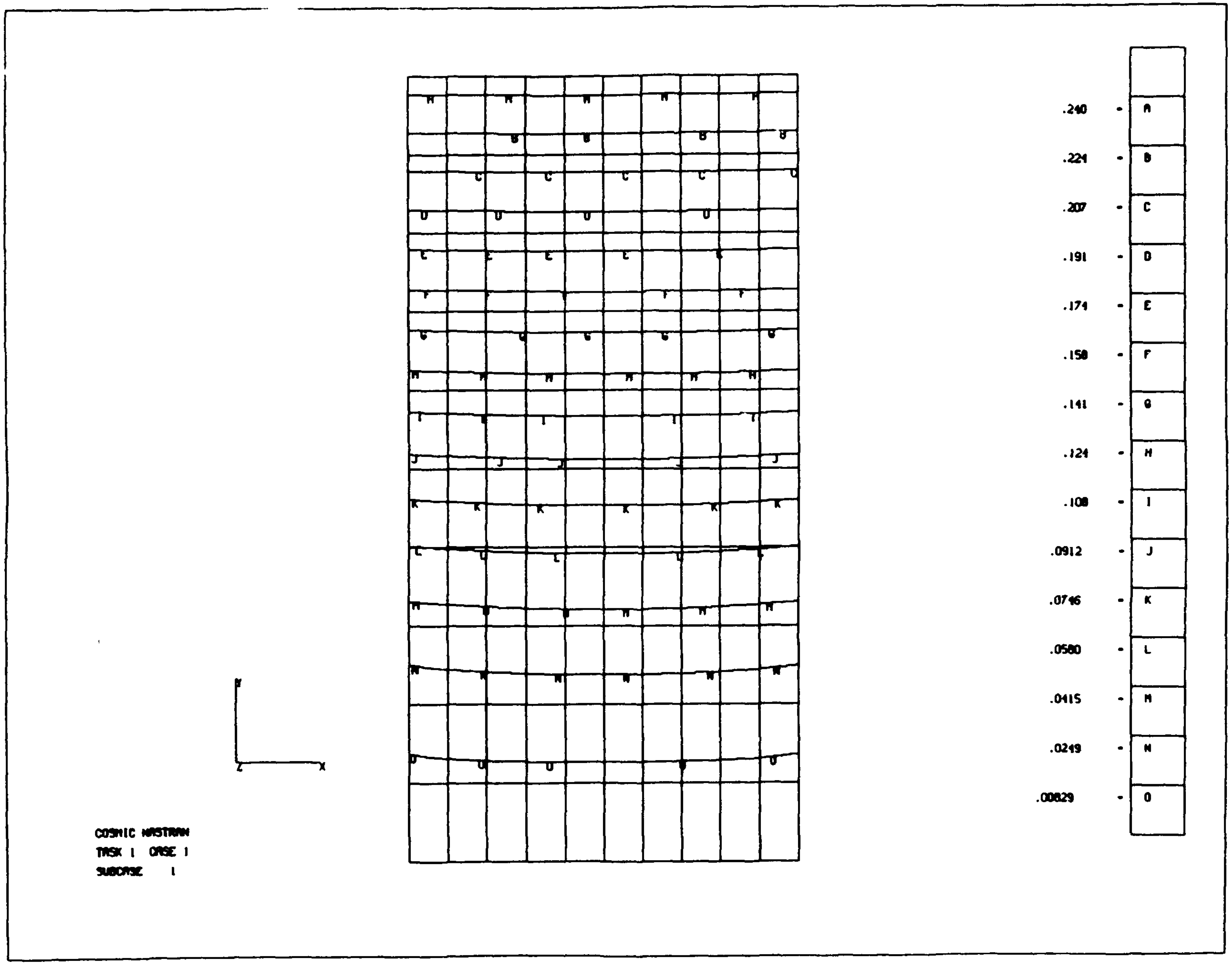


Figure 9. COSMIC NASTRAN Equivalent Displacement Contours for Case 1.

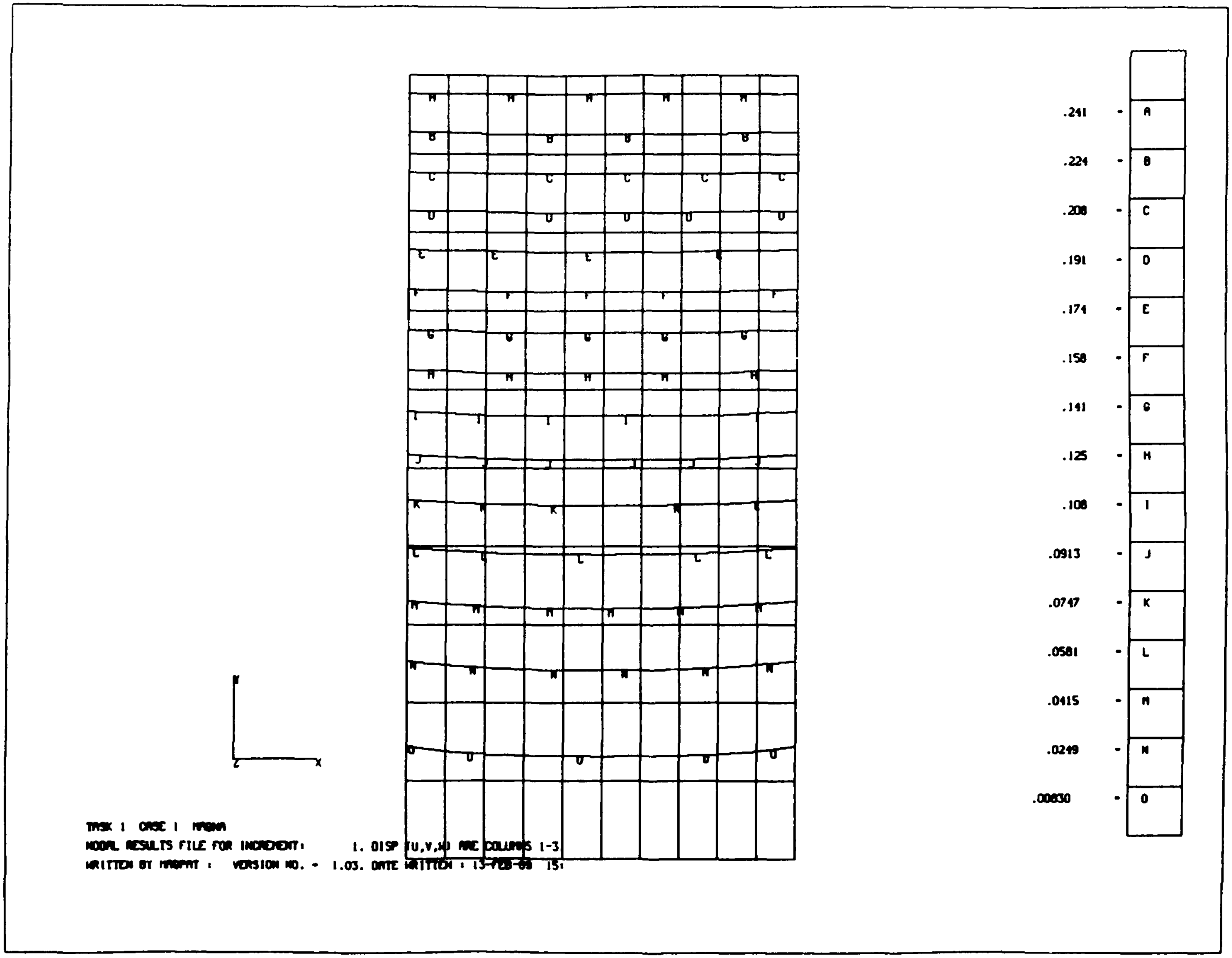


Figure 10. MAGNA Equivalent Displacement Contours for Case 1.

results are presented in Figures 11 to 13. The maximum displacements and equivalent stresses for all of the Task 1 analyses are summarized in Table 3. The theoretical values (Equations 14 and 16) are added for comparison.

The displacement results show good agreement for all of the finite element analyses. The simplified theoretical displacements are slightly higher than the finite element results by approximately 2 percent for all the analyses. The plate width has little, if any, effect on the maximum displacement as the beam theory presupposes. Finite element displacements vary with the cube of the thickness and the fourth power of the length.

MAGNA stresses agree well with the theoretical values especially for shorter and thicker plate configurations. ADINA and COSMIC NASTRAN maximum von Mises stresses are approximately 10 percent lower than the theoretical and MAGNA results. ADINA and COSMIC NASTRAN stress results are computed at the element centroid. Nodal values are determined by averaging the element results from all the elements to which the node is connected. Since the nodes along the support are attached only to elements across the plate width and since the behavior is not a function of that dimension, the stress values at the fixed edge are actually the stresses at the centroid of the elements nearest the support. If the theoretical expression for the stress is used, these centroid stress can be related to the maximum stress at the support by:

$$\sigma_{max} / \sigma_{centroid} = L^2 / (L - y)^2 \tag{17}$$

For all of the Task 1 models, the centroid of the elements located nearest the wall is located 5 percent of the distance from the support to the free edge of the plate. Therefore, the previous equation reduces:

$$\sigma_{max} / \sigma_{centroid} = L^2 / (L - .05 L)^2 = 1.108 \tag{18}$$

Equation 18 shows that the ten percent difference between the maximum ADINA and COSMIC NASTRAN stresses and the MAGNA counterparts is due to the integration and extrapolation methods used in the finite element analysis or in the reduction of the results. Some knowledge of the stress evaluation and extrapolation procedures is needed in order to interpret the graphical results.

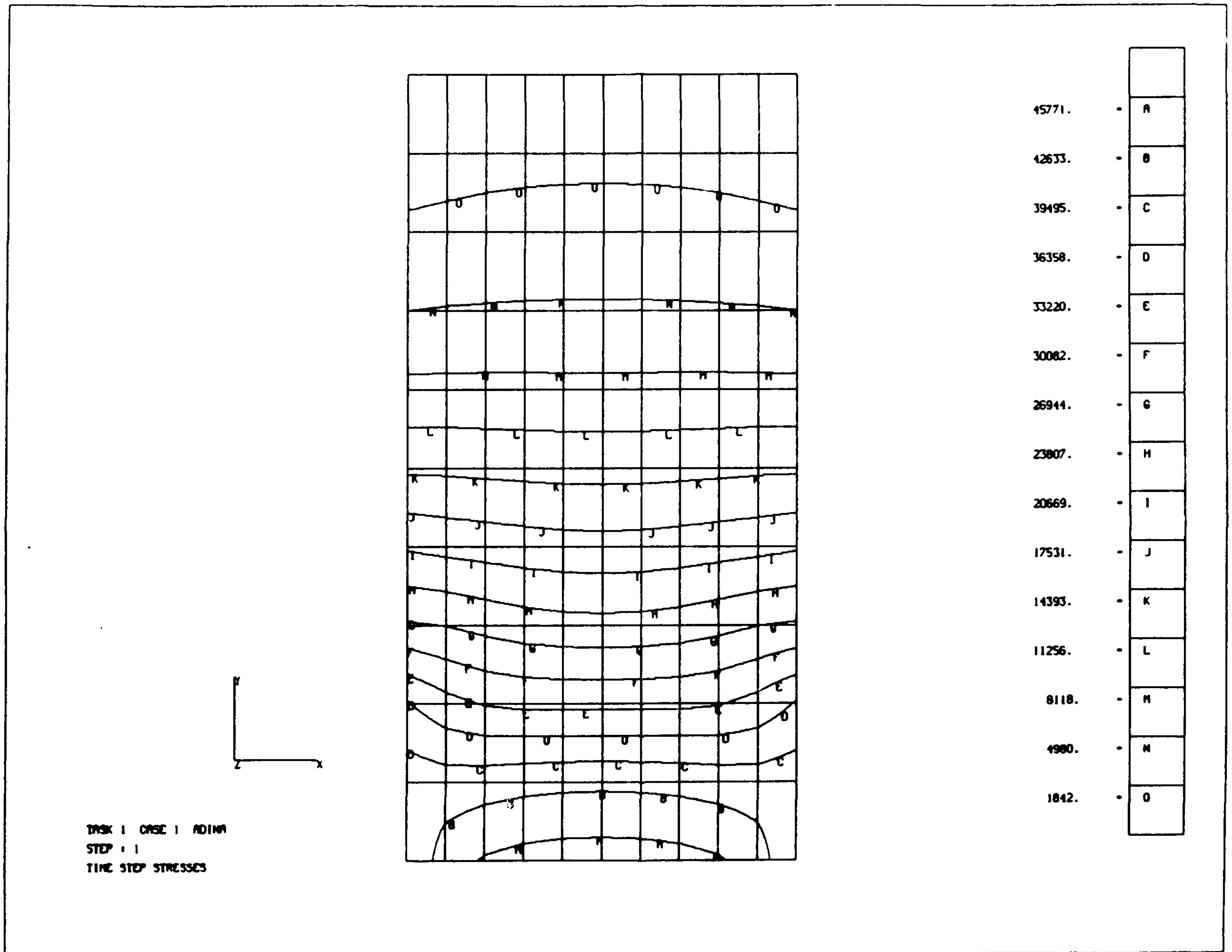


Figure 11. ADINA Equivalent Stress Contours for Case 1.

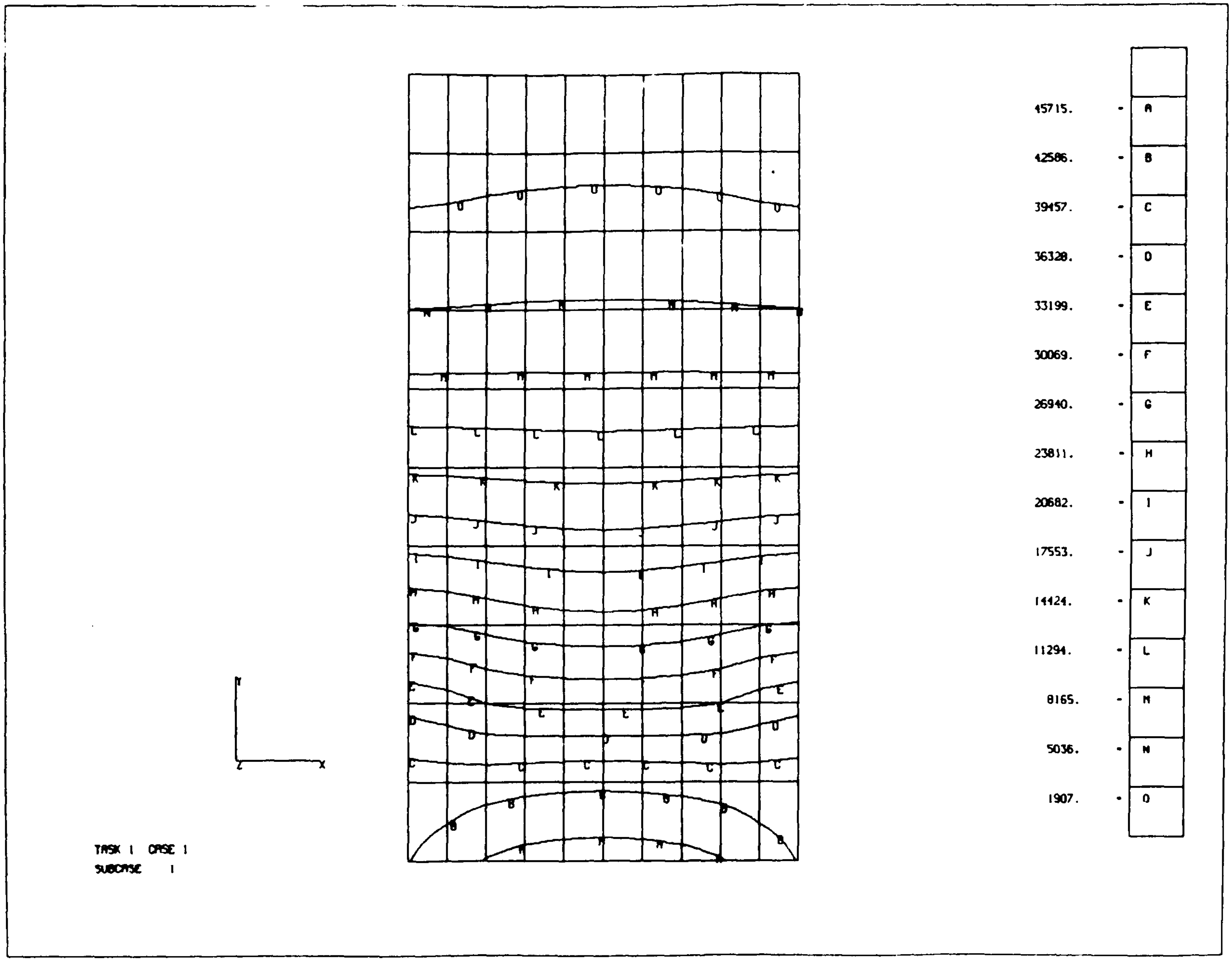


Figure 12. COSMIC NASTRAN Equivalent Stress Contours for Case 1.

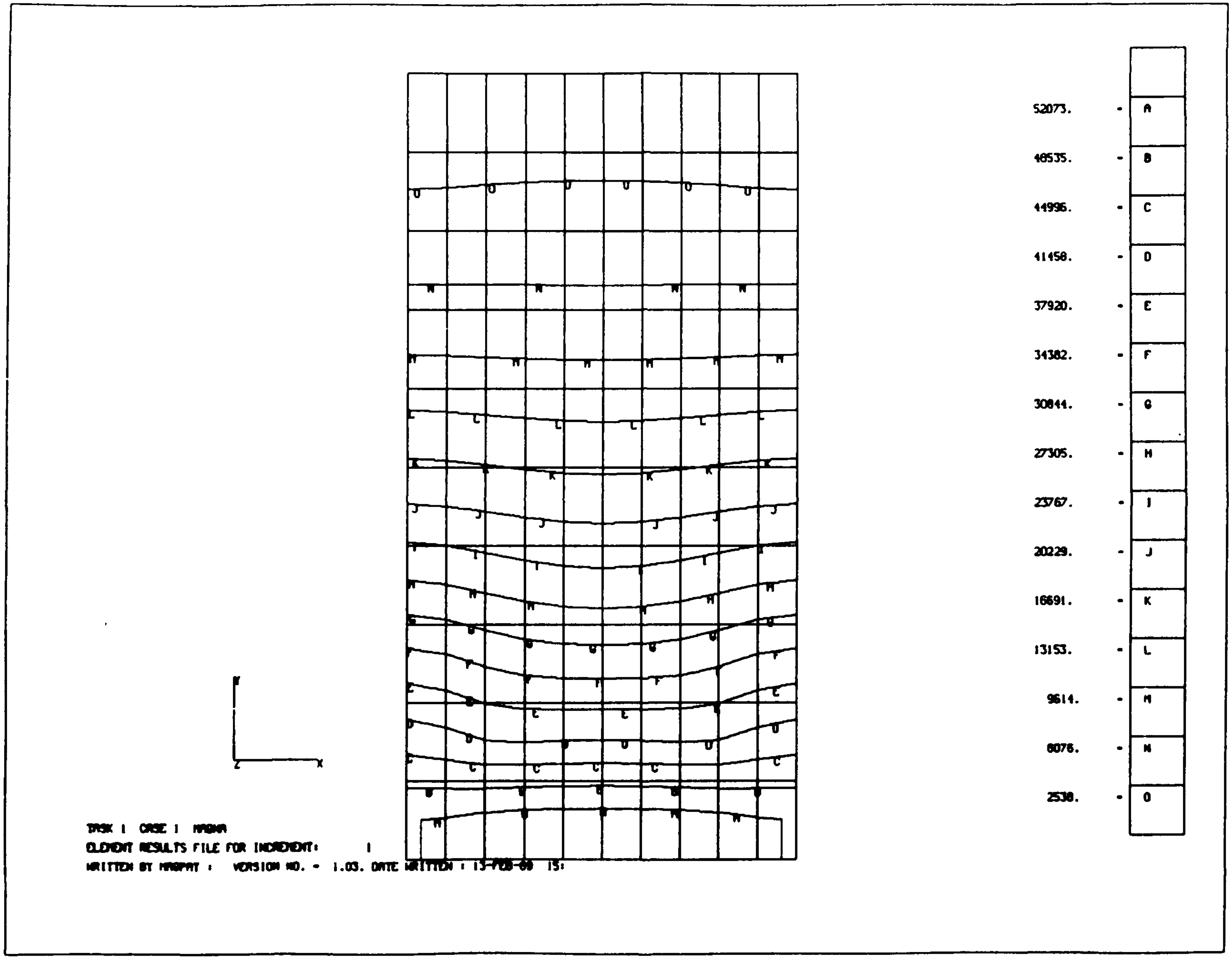


Figure 13. MAGNA Equivalent Stress Contours for Case 1.

TABLE 3

TASK 1 FINITE ELEMENT ANALYSIS RESULTS

Length (in)	Width (in)	Analysis Code	3 Plies w_{max} (in)	3 Plies σ^{vm}_{max} (Ksi)	5 Plies w_{max} (in)	5 Plies σ^{vm}_{max} (Ksi)	7 Plies w_{max} (in)	7 Plies σ^{vm}_{max} (Ksi)
2.	1.	Theory	0.254	53.3	0.055	19.2	0.020	9.80
		ADINA	0.249	47.3	0.054	17.0	0.020	8.70
		COSMIC	0.250	47.3	0.054	17.0	0.020	8.70
		MAGNA	0.249	53.8	0.054	19.3	0.020	9.80
2.	2.	Theory	0.254	53.3	0.055	19.2	0.020	9.80
		ADINA	0.248	46.7	0.054	16.8	0.020	8.60
		COSMIC	0.249	46.6	0.054	16.8	0.020	8.60
		MAGNA	0.248	52.5	0.054	18.8	0.020	9.50
2.	4.	Theory	4.06	213.	0.878	76.8	0.320	39.2
		ADINA	3.98	189.	0.861	68.2	0.313	34.8
		COSMIC	3.99	189.	0.863	68.1	0.314	34.7
		MAGNA	3.98	234.	0.861	77.9	0.314	39.4
4.	4.	Theory	4.06	213.	0.878	76.8	0.320	39.2
		ADINA	3.97	187.	0.858	67.2	0.313	34.3
		COSMIC	3.98	187.	0.863	67.2	0.313	34.3
		MAGNA	3.97	218.	0.857	76.0	0.313	38.5

SECTION 8

CONCLUSIONS AND RECOMMENDATIONS

As a result of this investigation, the following conclusions regarding the in-house structural analysis capabilites of the Aero Power and Propulsion Laboratory are formulated:

1. CRAY finite element analyses (ADINA, COSMIC NASTRAN, and MAGNA) have been completed using models constructed and post-processed using PATRAN on POTCM2.
2. PATRAN has been successfully used to model cantilevered composite flat plate geometries. Using PATRAN input data files, parametric variations are easily modeled. Extension to other problems and configurations is straight forward.
3. All three pre-analysis translators (PATADI, PATCOS, and PATMAG) are operational on POTCM2.
4. File transfer between POTCM2 and the CRAY VAX using the DECnet/TELNET protocol operates satisfactorily.
5. The procedure, CRAY_SUBMIT, to generate CRAY Job Control Language necessary to perform finite element analyses on the CRAY, functions correctly for static analyses using ADINA, COSMIC NASTRAN, and MAGNA.
6. Modifications to COSPAT permit COSMIC NASTRAN analyses to be performed on the CRAY for single load case static analyses.
7. The post-analysis translators ADIPAT and MAGPAT are operational on POTCM2.
8. Post-processing of finite element results in PATRAN is supported with several output devices including the Tektronix 4396D color printer and the LN03 laser printer.
9. Results for the Task 1 analyses showed good agreement between the theoretical behavior and the three finite element codes ADINA, COSMIC NASTRAN, and MAGNA.

10. Apparent discrepancies in the effective stress contour plots for different finite element analysis code results of the same problem are a consequence of the smoothing and extrapolation methods used in the finite element program or in the data reduction. Knowledge of the extrapolation and integration procedures is essential to correct interpretation of results.

The recommendations below indicate efforts for future tasks and suggestions to improve existing software and operations.

1. Support for the PSHELL and CQUAD4 data types in PATCOS will eliminate the need to add such data manually.

2. Additional modifications to COSPAT will be needed to permit several COSMIC NASTRAN load cases or normal modes to be analyzed in the same CRAY batch job.

3. Modifying PATMAG to convert the PATRAN 16-node solid element into MAGNA layered shell elements would eliminate the need to generate 20-node PATRAN solid elements and constrain the four extraneous midside nodes.

4. Modifications in PATADI are desirable to allow translation of shell material types other than isotropic and automatic conversion of all zero prescribed displacements into boundary conditions.

5. Error messages added to COSMIC NASTRAN would prevent inappropriate usage of the PSHELL card, the limitations of which are described in the documentation but not enforced on execution.

6. COSPAT and PATCOS are considered by PDA Engineering to be mature software which will not be updated. Any future changes to PATRAN or COSMIC NASTRAN will not be supported by the translators unless efforts are initiated in-house to adapt the codes.

REFERENCES

1. ________, ADINA (Automatic Dynamic Incremental Nonlinear Analysis) User's Manual, Report ARD 84-1, ADINA R&D, Inc., Watertown, Massachusetts, December 1984.

2. ________, NASTRAN User's Manual; Volume 1, NASA SP-222(08), June 1986.

3. Brockman, R. A., MAGNA (Materially and Geometrically Nonlinear Analysis) Part I - Finite Element Analysis Manual, AFWAL-TR-82-3098, Air Force Wright Aeronautical Laboratories, Wright Patterson Air Force Base, Ohio, December 1982.

4. ________, PATRAN PLUS User's Manual, PDA Engineering, Costa Mesa, California, 1987.

5. ________, PAT/ADINA Application Interface, PDA Engineering, Costa Mesa, California, 1987.

6. Jones, R., Mechanics of Composite Materials, McGraw-Hill, New York, 1975.

7. ________, PAT/COSMIC NASTRAN Application Interface, PDA Engineering, Costa Mesa, California, 1987.

8. Venkayya, V. B., "QUAD4 Seminar Notes", NASTRAN User's Colloquium, Wright-Patterson Air Force Base, April 1988.

9. Brockman, R. A., Held, T. W., and Dominic, V. G., MAGNA-to-PATRAN Data Interface User's Guide, UDR-TR-86-84, University of Dayton Research Institute, Dayton, Ohio, July 1986.

10. ________, COS Version 1 Reference Manual, SR-0011, CRAY Research, Inc., Mendota Heights, Minnesota, 1986.

11. Gere, J. M. and Timoshenko, S. P., Mechanics of Materials, PWS Publishers, Boston, 1984.

APPENDIX A

DECnet/TELNET FILE TRANSFER PROTOCOL

The combined DECnet/TELNET file transfer protocol is necessary to transmit data between the CRAY VAX and POTCM2. DECnet links POTCM2 to POHOST which is an ULTRIX system (case sensitive) in Building 18. POHOST and the CRAY VAX are active nodes on the ARPANET system. ARPANET is a portion of the Defense Data Network which is a nationwide network of military computer systems. File transfer on the ARPANET system is performed using TELNET commands.

Files can be transferred both to and from POTCM2, but the directives must always be initiated on POTCM2 because the CRAY VAX does not support DECnet. Also, only formatted (ASCII) files will transfer properly. The sections below describe the procedures required to send or retrieve data between POTCM2 and the CRAY VAX.

1. File transfer from POTCM2 to the CRAY VAX

$ COPY file.ext pohost"vlcc!username password"::destfile.ext

where:

file.ext = POTCM2 file to be sent
username = CRAY VAX account name
password = CRAY VAX account password
destfile.ext = CRAY VAX file to be created

The file will arrive on the CRAY VAX slightly altered because the ULTRIX system handles file attributes differently than VMS systems. To restore the transferred file to its original form, type:

$ RUN [D870062.PUBLIC]FIX2VAR

while interactively logged into the CRAY VAX. This procedure will ask for the name of the file to convert, and a new file name for the repaired data.

2. File Transfer from the CRAY VAX to POTCM2

$ COPY pohost"vlcc!username password"::"file.ext" destfile.ext

where:

file.ext	=	CRAY VAX file to be retrieved
username	=	CRAY VAX account name
password	=	CRAY VAX account password
destfile.ext	=	POTCM2 file to be created

Again, the file will arrive on the POTCM2 slightly altered because the ULTRIX handles file attributes differently than VMS systems. Most of the editing and post-processing functions which are performed on POTCM2 will be unaffected. The one exception found to date is the post-processing of ADINA results from the CRAY VAX. If the file attributes are causing difficulties, then the original attributes can be restored on POTCM2 using:

$ @USERA:[TASK21]FIXFIL.COM

3. Interactive Access to the CRAY VAX

It is possible to interactively login to the CRAY VAX from POTCM2 using the same pathway as mentioned previously as indicated below.

(currently on POTCM2)

$ SET HOST pohost

(pohost login message)

login: **vlcc!**
Trying...
Connected to vlcc.wpafb.af.mil.
Escape character is '^]'.

(CRAY VAX login message)

Username: **account**
Password: **password**

(CRAY VAX system messages)

$ (now logged into the CRAY VAX)

APPENDIX B

CENTRAL FILE SYSTEM USAGE

By default, files on the Central File System (CFS) are stored in the native format of the front-end machine from which the files are written. Such native formats are machine dependent. Thus, a CDC local file written to the CFS using the native format cannot be read on the CRAY VAX and vice versa. Shared access to files requires that the files be stored in an ASCII form. The commands which follow indicate how to store and to retrieve ASCII formatted CFS files. Even if a CFS file is not needed on another front-end machine, it is recommended that all CFS files be stored in ASCII form to eliminate any confusion.

VAX to CFS: SWRITE **vaxfile.ext cfsfile** RFMT=VLB

CDC to CFS: FILE, **local1**, RT=Z, BT=C, FL=80
FILE, **local2**, RT=Z, BT=C, FL=80
FORMAT, INP=**local1**, OUT=**local2**
SWRITE, **local2**, **cfsfile**, RFMT=VLB, STYP=DIS, TTYP=ASC

CFS to VAX: SREAD **vaxfile.ext cfsfile**

CFS to CDC: FILE, **local3**, RT=Z
SREAD, **local3**, cfsfile

where: vaxfile.ext = CRAY VAX file name
local1, local2, local3 = CDC local file names
cfsfile = CFS file name

Note: 1. Variable length records are written on the VAX, CDC, and CFS.
2. If 132 column data is necessary then FL=132.
3. CDC format command copies the input file to the output file where each file can have different file structures.
4. CFS file names with periods must be enclosed in "$" when referenced on the CDC.

APPENDIX C

MAGNETIC TAPE DATA TRANSFER

The output files from some analyses may be too large to be returned directly to the CRAY VAX. In such an event the results are read onto the CDC and written to magnetic tape. If the tape is written in the proper format, the tape drive on POTCM2 can read the data properly.

The procedure is to write formatted (ASCII) files to magnetic tape. Binary file are not permissible due to machine dependencies. The tape specifications are:

1. ASCII format
2. Density = 6250
3. Unlabeled
4. Blocked (Block size = 3300 bytes)
5. Fixed length records (Record length=132)

The sample procedure which follows reads a CFS file and writes the file to magnetic tape using the FCOPY command. FCOPY converts the CDC display code into ASCII code with fixed length records. However, FCOPY will copy a file only until it finds the first CDC EOR or EOF mark. The CDC PACK command will remove all intermediate EOR and EOF marks from the file.

```
FILE,file1,RT=Z.
SREAD,file1,cfsfile1.
LABEL,TAPE,VSN=X00iii,D=GE,PO=W,F=S,LB=KU,CV=AS.
REWIND,TAPE.
PACK,file1.
REWIND,file1.
FCOPY,P=file1,N=TAPE,PC=DIS,NC=ASCFL,FL=132,LB=25.
ENQUIRE,F.
```

where:

```
filel    = CDC local file to be written to tape
cfsfilel = CFS file to be transferred
X00iii   = External tape label
```

On POTCM2, the tape can be read using the MTEXCH commands:

$ ALLOC **tapedrive** TAPE:
$ MOUNT /FOR /DEN=6250 /BLOCK=3300 TAPE:
$ MTEXCH
* TAPE: /REWIND
* **filel.dat** /VAR = TAPE: /FIXED /RECL=132
* EXIT
$ DIR/SIN
$ DISMOUNT TAPE:
$ DEALLOC TAPE:

where:

```
tapedrive = POTCM2 device name for tape drive
filel.dat = POTCM2 file name to be created
```

APPENDIX D

LN03 HARDCOPY OUTPUT OF PATRAN RESULTS

For documentation purposes, black and white images generally reproduce with higher quality or at least more economically than color graphics. PATRAN has the capability to create a file (PATRAN.HRD) that contains a description of the PATRAN graphics. This file can be processed to produce hardcopy output on another device such as the LN03 laser printer.

Before the desired plot is drawn on the screen in PATRAN, issue the command "SET,HARDCOPY,ON". This causes all the plot information which follows to be written to PATRAN.HRD in the current subdirectory on POTCM2. Graphic images will continue to be added to the hardcopy file until the command "SET,HARDCOPY,OFF" is issued or the current PATRAN session is ended.

A FORTRAN program was written to convert the PATRAN graphics directives into DISSPLA (a product of ISSCO, San Diego, Ca.) subroutine calls. To generate a DISSPLA format hardcopy file, type:

$ RUN USERA:[FIELMAN.DISSPLA]DISSLA_DRIVER

and select the HARDDISS option during execution. The user selects the device that will be used to present the plot or plots. If the LN03 is selected, the file STD00001.DAT containing the plot information will be written in the current directory on POTCM2. This file can be sent to the printer using:

$ PRINT /NOFEED /DELETE STD0001.DAT

The NOFEED option is required to suppress line feed directives in the first column of the file. The DELETE option is recommended due to the large size of the STD0000.DAT file and the ease with which it can be recreated.

APPENDIX E

CQUAD4 ELEMENT DEFINITION IN COSMIC NASTRAN

Input Data Card COQUAD4   Quadrilateral Element Connection

Description: Defines a quadrilaterial plate element (QUAD4) of the structural model. This is an isoparametric membrane-bending element, with variable element thickness, layered composite material and thermal analysis capabilities.

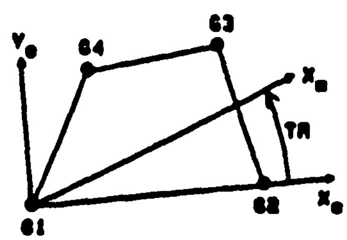
Format and Example:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	TM	ZO	abc
CQUAD4	101	17	1001	1005	1010	1024	45.0	0.01	ABC

+bc			T1	T2	T3	T4			
+BC			0.03	0.125	0.05	0.04			

Field	Contents
EID	Element identification number (Integer > 0)
PID	Identification number of a PSHELL entry (Default is EID) (Integer > 0) For composites, see Remark 5.
Gi	Grid point identification numbers of connection points (Integer > 0)
ZO	Offset of the element reference plane from the plane of grid points (Real or blank, see Remark 3 for default)
TM	Material property orientation specification (Real or blank; or 0 ≤ Integer < 1,000,000). If Real or blank, specifies the material property orientation angle in degrees. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value.
Ti	Membrane thickness of element at grid points Gi (Real or blank, see Remark 4 for default).

Remarks: 1. The QUAD4 geometry, coordinate systems and numbering are shown in the figure below:



2. Element identification numbers must be unique with respect to all other element identification numbers.

(Continued)

3. The material coordinate system (TM) and the offset (ZO) may also be provided on the PSHELL entry. The PSHELL data will be used if the corresponding field on the CQUAD4 entry is blank.

4. The Ti are optional, if not supplied they will be set to the value of T specified on the PSHELL entry. In such cases, the continuation entry is not required.

5. For composites, a PCOMP, PCOMP1, PCOMP2 card can be used instead of a PSHELL card.

APPENDIX F

PSHELL PROPERTY DEFINITION IN COSMIC NASTRAN

Bulk Data Entry PSHELL   Shell Element Property

Description: Defines the membrane, bending, transverse shear, and coupling properties of the QUAD4 shell element.

Format and Example:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T³	MID3	TS/T	NSM	abc
PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	ABC

+bc	Z1	Z2	MID4	MCSID	SCSID	Z0			
+BC	+.95	-.95		0	0	0.01			

Field	Contents
PID	Property identification number (Integer > 0)
MID1	Material identification number for membrane (Integer > 0 or blank)
T	Default value for membrane thickness (Real > 0.0)
MID2	Material identification number for bending (Integer > 0 or blank)
12I/T³	Bending stiffness parameter (Real or blank, default = 1.0)
MID3	Material identification number for transverse shear (Integer > 0 or blank, must be blank unless MID2 > 0)
TS/T	Transverse shear thickness divided by membrane thickness (Real or blank, default = .833333)
NSM	Nonstructural mass per unit area (Real)
Z1,Z2	Fiber distances for stress computation. The positive direction is determined by the righthand rule and the order in which the grid points are listed on the connection entry. (Real or blank, defaults are -T/2 for Z1 and +T/2 for Z2).
MID4	Material identification number for membrane-bending coupling (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0, may not equal MID1 or MID2)
MCSID	Identification number of material coordinate system (Real or blank, or (Integer ≥ 0) (See Remark 11)
SCSID	Identification number of stress coordinate system (Real or blank, or (Integer ≥ 0) (See Remark 11)
Z0	Offset of the element reference plane from the plane of grid points. (Real or blank, default = 0.0) (See Remark 12)

(Continued)

PSHELL (Continued)

rks: 1. All PSHELL property entries must have unique identification numbers.

2. The structural mass is computed from the density using the membrane thickness and membrane material properties.

3. The results of leaving any MID field blank are:

MID1	No membrane or coupling stiffness
MID2	No bending, coupling, or transverse shear stiffness
MID3	No transverse shear flexibility
MID4	No membrane-bending coupling

4. The continuation entry is not required.

5. Structural damping, when needed, is obtained from the MID1 material.

6. The MID4 field should be left blank if the material properties are symmetric with the middle surface of the shell.

7. For structural problems, PSHELL entries may reference MAT1, MAT2 or MAT8 material property data.

8. If the transverse shear material, MID3, references MAT2 data, then G33 must be zero. If MID3 references MAT8 data, then G1,Z and G2,Z must not be zero.

9. For heat transfer problems PSHELL entries may reference MAT4 or MAT5 material property data.

10. If MCSID/SCSID is left blank (0.0) or is real, it is considered to be the angle of rotation of the X axis of the material/stress coordinate system with respect to the X axis of the element coordinate system in the XY plane of the latter. If integer, the orientation of the material/stress x-axis is along the projection of the x-axis of the specified coordinate system onto the x-y plane of the element system. The value of MCSID is the default value for the TM field on the CQUAD4 Bulk Data entries.

11. The value of ZO is the default value for the corresponding field on the CQUAD4 Bulk Data entries.

APPENDIX G

MAT8 MATERIAL DEFINITION IN COSMIC NASTRAN

Input Data Card MAT8 Orthotropic Plate Material Property Definition

Description: Defines the material property for an orthotropic material for plate elements.

Format and Example:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	abc
MAT8	299	32.+6	4.2+5	0.33	2.9+6			0.042	ABC
+bc	A1	A2	TREF	XT	XC	YT	YC	S	def
+BC	14.-6	2.3-5	175.						DEF
+ef	GE	F12							
+EF	2.5-4								

Field	Contents
MID	Material identification number (Integer > 0)
E1,E2	Modulus of elasticity in the material x and y directions (Real ≠ 0.0)
NU12	Poisson's Ratio (Real) (See Remark 5)
G12	Linear In-plane shear modulus (Real > 0.0)
G1Z	Transverse shear modulus for shear in X-Z plane (Real)
G2Z	Transverse shear modulus for shear in Y-Z plane (Real)
RHO	Mass density (Real)
A1,A2	Thermal expansion coefficients in the material x and y directions (T, Real > 0.0)
TREF	Thermal expansion reference temperature (XC, Real)
XT,XC	Allowable stresses/strains in tension and compression, respectively, in the material x direction. Required if failure index calculation is desired. (XT, Real > 0.0) (XC, Real) (Default value for XC is XT) (See Remark 3)
YT,YC	Allowable stresses/strains in tension and compression, respectively, in the material y direction. Required if failure index calculation is desired. (YT, Real > 0.0) (YC, Real) (Default value for YC is YT) (See Remark 3)
S	Allowable stress/strain for in-plane shear (Real > 0.0) (See Remark 3)
GE	Structural damping coefficient (Real)
F12	Tsai-Wu interaction term (Real) (See Remark 4)

(Continued)

<u>MAT8</u> (Continued)

Remarks:

1. Material coordinate systems are defined by the plate element connection entries on the CQUAD4 card.

2. The stress-strain relationship defined by this data is:

$$\begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} 1/E1 & -NU12/E1 & 0 \\ -NU12/E1 & 1/E2 & 0 \\ 0 & 0 & 1/G12 \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} + (T-TREF) \begin{Bmatrix} A1 \\ A2 \\ 0 \end{Bmatrix}$$

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} G1Z & 0 \\ 0 & G2Z \end{bmatrix} \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix}$$

3. Fields XT, XC, YT, YC and S are used only for composite materials when failure calculations are requested with PCOMP, PCOMP1 or PCOMP2 Bulk Data entries. Allowables represent stresses except when the maximum strain failure theory is used.

4. The F12 field is used only for composite materials when the Tsai-Wu failure theory is used and failure calculations are requested.

5. NU12 is Poisson's Ratio (ϵ_1/ϵ_2 for uniaxial loading in 1-direction). Note that NU21 = ϵ_1/ϵ_2, uniaxial loading in 2-direction, is related to NU12, E1 and E2 by the relationship, (NU12) (E2) = (NU12) (E1).

APPENDIX H

COSPAT MODIFICATIONS

COSMIC NASTRAN results on the CRAY are written to the binary OUTPUT2 file. If COSPAT is executed on the CRAY immediately after the COSMIC NASTRAN analysis, the OUTPUT2 file can be converted into ASCII PATRAN results files. These ASCII files are machine independent and can be returned to POTCM2 for post-processing.

Unfortunately, no CRAY version of COSPAT is supported by PDA Engineering, the authors of PATRAN. A CDC version of the COSPAT source code was received from PDA and converted to run on the CRAY. Support for the QUAD4 element was also added to the modified COSPAT code, because the QUAD4 element was added to COSMIC NASTRAN after the last version of COSPAT was released.

The comments below outline the modifications that were required to port the CDC version of COSPAT to the CRAY and incorporate the QUAD4 element.

1. CRAY conversion

- CDC dependent time and date subroutines were replaced with CRAY counterparts.

- Character string assignments were modified to remove:

  string(a:b) = string (c:d)

  which are not permitted in CRAY FORTRAN.

2. QUAD4 addition

- Internal element type for QUAD4 element was changed from 33 to 64.

3. Multiple load cases or modes (not yet completed)

- CRAY intrinsic function will be used to change the file alias during execution so that the results for each load case or mode will be written to a unique file.

4. Sample COSMIC NASTRAN and COSPAT CRAY batch job

- The procedure below performs a COSMIC NASTRAN analysis on the CRAY immediately followed by the COSPAT post-analysis translation. The file listed below was generated by the procedure CRAY_SUBMIT on the CRAY VAX.

Table H-1.

CRAY Job Submittal Procedure for COSMIC NASTRAN Analyses

```
JOB,JN=cosmic,T=200,US=P890028,MFL,SSD=1.
ACCOUNT,AC=P890028,APW=crayapw,UPW=crayupw.
*. CRAY JOB SUBMISSION
*. COSMIC NASTRAN FINITE ELEMENT ANALYSIS
REWIND,DN=$OUT.
ACCESS,DN=NPROC,ID=COSMIC88,OWN=A860076.
ACCESS,DN=CPMEXE,PDN=CPMEXE.
LIBRARY,DN=NPROC:*.
COPYF,I=$IN,O=CPINP.
ASSIGN,A=FT11,DN=OT2.
*.
*. Execute COSMIC NASTRAN
*.
NASTRAN.
DS.
REWIND,DN=OT2.
COPYF,I=OT2,O=OUTPUT2.
RELEASE,DN=OT2.
REWIND,DN=OUTPUT2:CPINP.
ASSIGN,A=FT05,DN=CPINP.
ASSIGN,A=FT04,DN=OUTPUT2.
ASSIGN,A=FT11,DN=SUBCA1D.
ASSIGN,A=FT13,DN=SUBCA1E.
*.
*.  Execute COSPAT
*.
CPMEXE.
*.
*. Dispose standard output and results files
*.
DISPOSE,DN=SUBCA1D,SDN=SCA1D,MF=CB,DF=CB,DC=ST,^
TEXT='USER,P890028,cdcbpw.CTASK.FILE,SCA1D,BT=C,RT=Z,FL=80.'^
'FILE,CA1D,BT=C,RT=Z,FL=80.FORM,INP=SCA1D,OUT=CA1D.'^
'SWRITE,CA1D,cosmicdr,RFMT=VLB,STYP=DIS,TTYP=ASC.'.
DISPOSE,DN=SUBCA1E,SDN=SCA1E,MF=CB,DF=CB,DC=ST,^
TEXT='USER,P890028,cdcbpw.CTASK.FILE,SCA1E,BT=C,RT=Z,FL=80.'^
'FILE,CA1E,BT=C,RT=Z,FL=80.FORM,INP=SCA1E,OUT=CA1E.'^
'SWRITE,CA1E,cosmicer,RFMT=VLB,STYP=DIS,TTYP=ASC.'.
DISPOSE,DN=$OUT,SDN=OUT,MF=CB,DF=CB,DC=ST,^
TEXT='USER,P890028,cdcbpw.CTASK.FILE,OUT,BT=C,RT=Z,FL=132.'^
'FILE,COUT,BT=C,RT=Z,FL=132.FORM,INP=OUT,OUT=COUT.'^
'SWRITE,COUT,cosmicot,RFMT=VLB,STYP=DIS,TTYP=ASC.'.
/EOF
```

Table H-1, continued

```
*.
*. COSPAT directives
*.
/ASCII:ON
1
1
OUTPUT2
Y
                  (COSPAT options)

Y


3
6
/EOF
*DECK COSMIC
                  (COSMIC NASTRAN input data deck
                   is inserted here by CRAY_SUBMIT)
/EOF
```

where:

cosmic.dat = CRAY VAX input data file

cosmicot = Standard output file on the CFS

cosmicdr = PATRAN displacement results file from COSPAT

cosmicer = FATRAN element results file from COSPAT

crayapw = CRAY account password

crayupw = CRAY user password

cdcbpw = CDC batch password

Note: 1. /ASCII:ON option in COSPAT insures that formatted (ASCII) PATRAN results files will be generated, not binary files which are the default. The formatted results file can then be returned to POTCM2 and post-processed in PATRAN.

2. To submit the CRAY JCL file listed above, type:

$ CRAY SUBMIT cosmic.job